FE ANALYSIS WITH EVALUATION OF STRESS INTENSITY FOR WELDED JOINTS OF VARISPRING COMPONENT

[1]Shaikh Mohammad Mohsin, [2]Ambadas, [3]Babu Reddy
[1]M.Tech Student, [2 & 3]Assistant Professor
[1, 2 & 3]Department of Machine Design, Center of Post Graduate Studies, VTU Kalaburagi, India

Abstract— This paper deals with FE analysis of varispring component The main objective is to evaluate Stress Intensity at the welded joints for varispring components. Because of Heavy Impact, Vibrations and Repetitive loading manual spotted welds loses its intensity and it may leads to decrease the stability as well as maintenance cost. During the analysis we have considered Irradiation at elevated Pressure, Force and Moments. As from the ASME standard sec VIII, Div I and II, verified with the material as well as actual loading conditions. The results obtained from the analysis are Stress, Displacement, Stress Intensity.

Index Terms— FE Analysis, Varispring component, stress intensity, welded joints, Pressure, ASME Standards etc.

I. INTRODUCTION

The modern mechanized world depends largely on several mechanical components to survive each day. One such component is the Varispring, The varispring components consists Torque limiter machines, Pin Hole welding machines, which is mostly used in Automobile Industries. Nowadays usage to sustain loading in numerous fields ranges from town water distribution networks to sub ocean oil transportation to cryogenic applications. Therefore it was deemed to study such a crucial component in today's industrialized world and thus this study was taken up to analyze the various failure criterions of welded neck for varispring components. To optimize their performance in service and also to come up with standards and procedures in their manufacturing and integration with larger systems. The pressurized components of the shell are designed to be in accordance with a design code such as ASME VIII 2013 edition.

Fig1. Typical Varispring component

Generally varispring components consist of flange and pipe settings connected by the welded joints. The pipe and flange can be connected by slip on, screwed or welded. They can be butt welded or fillet welded depending upon the requirement. The weld neck flanges are most adequate amongst the types of flanges due to their reliability under different loading condition. The pipe and flange are selected such that their nominal pipe sizes are equal, to get same diameter and thickness at weld point.

II. METHODOLOGY.

- Each part of the component is created and assembled in CATIA software. For Numerical analysis the worst loading conditions are employed based on available data. Solver used for analysis is Ansys. Stress intensities and deformation obtained from results are checked for limit by design. FEA results are validated with Experimental results.

III. MODELLING, MATERIAL SELECTION

Varispring model consist of B16.5 flange of nominal diameter 3inch and pipe schedule 80 connected by weld1, and the pipe is welded to base plate by weld 2. Whole geometry is created in CATIA software.

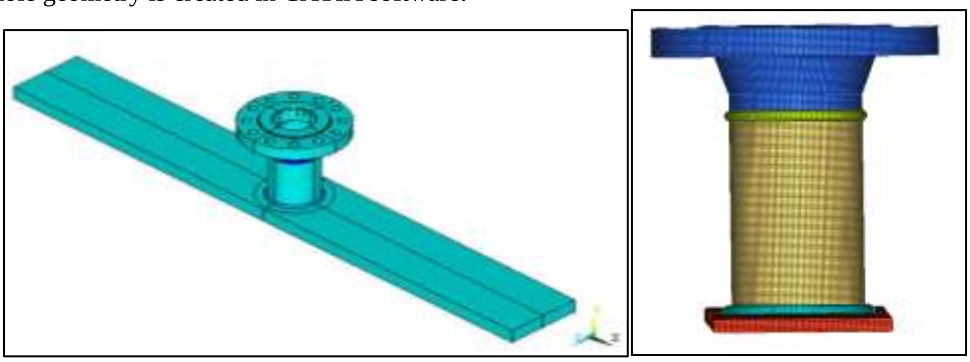


Fig.2 (a)Varispring component Model (b) Meshed model

Meshing have been carried out in hypermesh software. The model geometry is imported in hypermesh software. Material and material properties such as Youngs modulus and poissons ratio respectively assigned to the parts as shown in table 1. Element types are assigned respectively. The elements taken for flange and weld is 10 noded tetrahedral element solid 92 and for shell and base plate solid 45, 8 noded brick element. Total No of Nodes are found to be 976300 and total number of Elements is 706300.

Selection of Materials

Weld neck flanges are generally manufactured in all steel material grading, including carbon weld neck flanges, stainless steel weld neck flanges and alloy weld neck pipe flanges. Grade 304 stainless steel is most widely used grade steel, due to its resistance to corrosion, machinability and weld-ability characteristics. Its chemical composition includes 18% chromium 8% Nickel and 2% magnesium. It is extensively used in welding application because of low carbon content about .02%. .It is used to manufacturing flanges pipes, pressure vessel valves, fasteners and fixings etc. As from the ASME standards following materials are selected with their Strength, Toughness, Hardness, Ductility etc. parameters to achieve prescribed Quality.

Table.1 Material Properties

Material	**SA 240 Gr. 304**	**SA 312 TP 304**	**SA 182 F 304**	**Steel**
Property	**Bottom Header Plate**	**Vertical Shell**	**Flange**	**Weld 1 and 2**
Modulus of Elasticity in MPa	176000	176000	176000	210000
Poisson's Ratio	0.3	0.3	0.3	0.3
Yield strength Mpa	170	170	170	240
Density Kg/mm^3	7.85E-06	7.85E-06	7.85E-06	785E-06

Boundary condition

All sides of base plate are fixed and the loads and moments along with pressure is applied On the basis of worst loading conditions following load cases are defined.

1. Internal Design Pressure 5 MPa + Flange loads
2. Internal Test Pressure 7.65 MPa+ Flange loads
3. (Full vacuum) External Pressure 0.103MPa + Flange loads

Fx = 12000 N Mx = 6420*1000 N-mm
Fy = 15090 N My = 9150*1000 N-mm
Fz =15090 N Mz = 4890*1000 N-mm

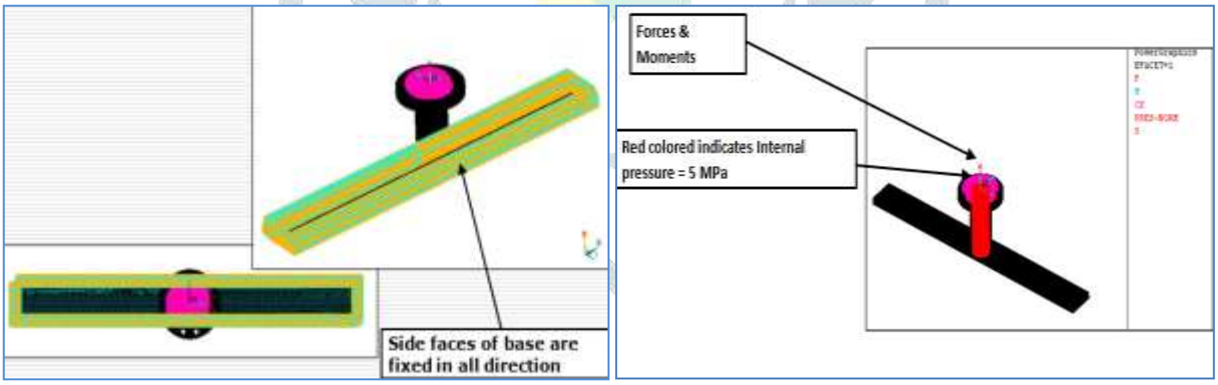


Fig.3 Boundary condition

IV. FE ANALYSIS

Load case 1: Operating loading: Internal Design Pressure 5 MPa + Flange loads

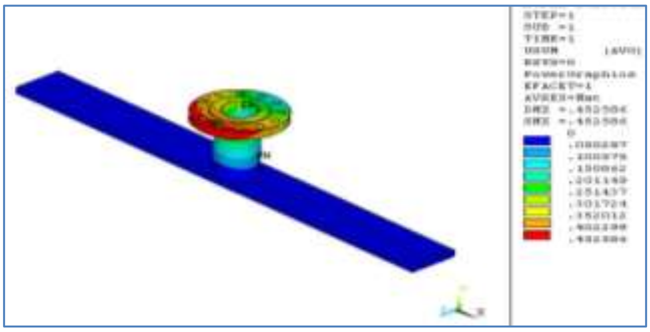


Fig.4 Vector deformation

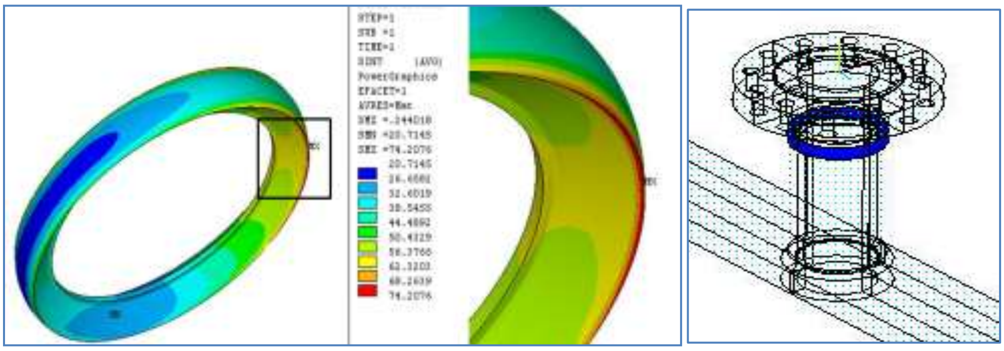


Fig.5 Stress intensity at weld 1

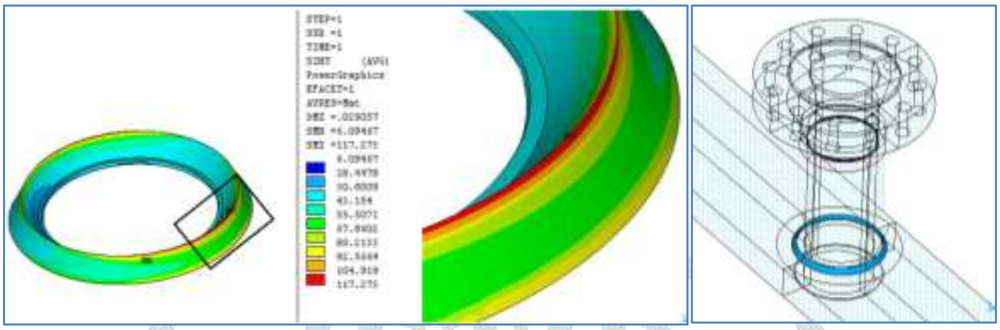


Fig.6 Stress intensity at weld 2

Load case 2: Maximum condition: Internal Test Pressure 7.65 MPa+ Flange loads

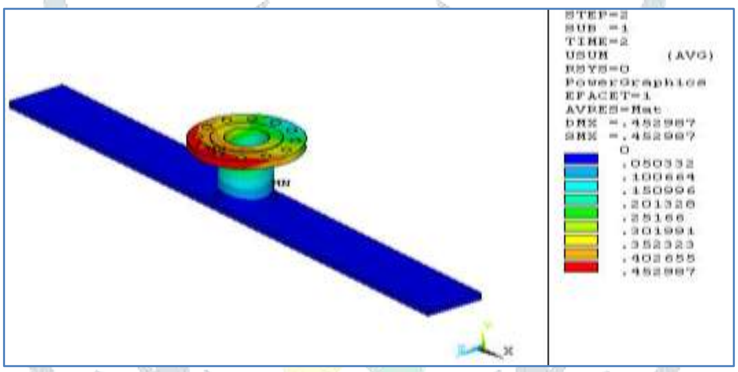


Fig 7 vector deformation

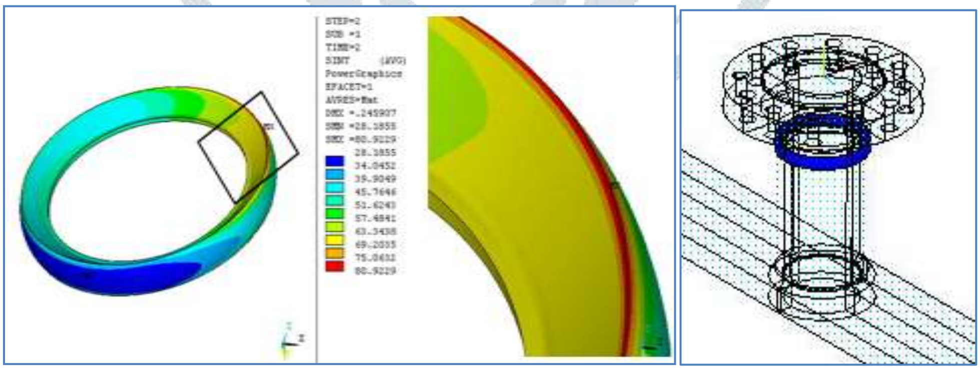


Fig. 8 stress intensity at weld 1

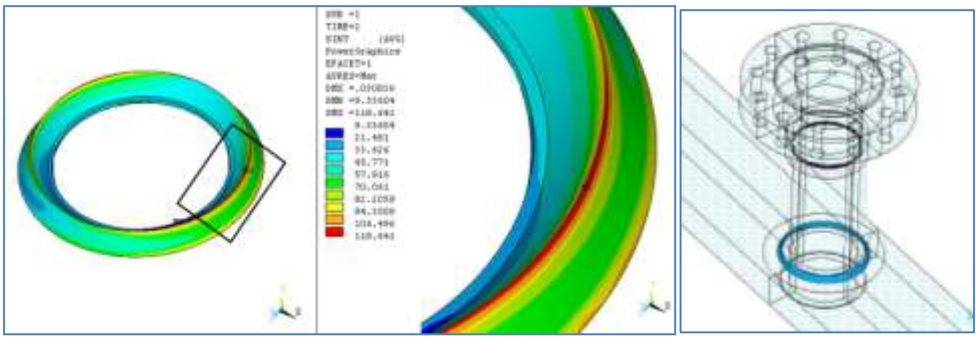


Fig.9 Stress intensity at weld 2

Load case 3: Shutdown condition: External Pressure 0.103MPa + Flange loads

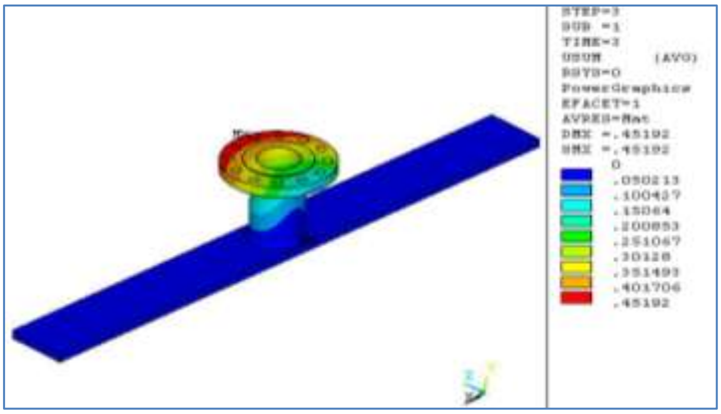


Fig .10 vector deformation

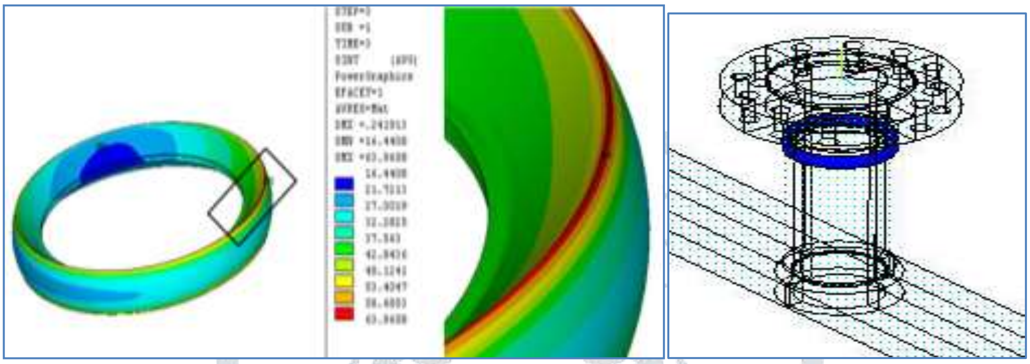


Fig. 11 Stress intensity a weld 1

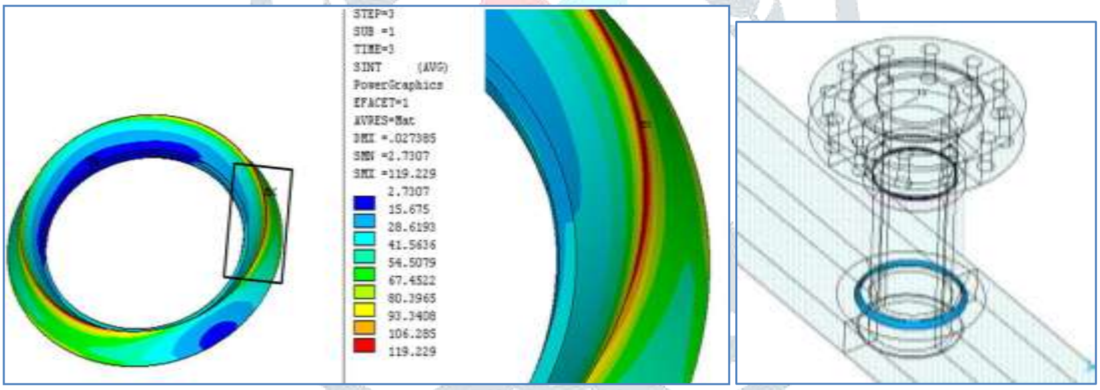


Fig12. Stress intensity at weld 2

V. FEA RESULTS

Sr No	Load Case	Deformation (mm)	Stress Intensity	
			Weld1 (MPa)	Weld 2 (Mpa)
1	Operating loading	0.4525	74.2076	117.27
2	Maximum condition	0.4529	80.9229	118.641
3	Shutdown condition	0.4519	63.9658	119.229

Table.2 FEA Results

The maximum deformation occurs in maximum loading condition in X direction**.** Stress intensities obtained for weld 1 & weld 2 obtained for all the load cases are within the yield stress of 240 Mpa. Therefore design is safe.

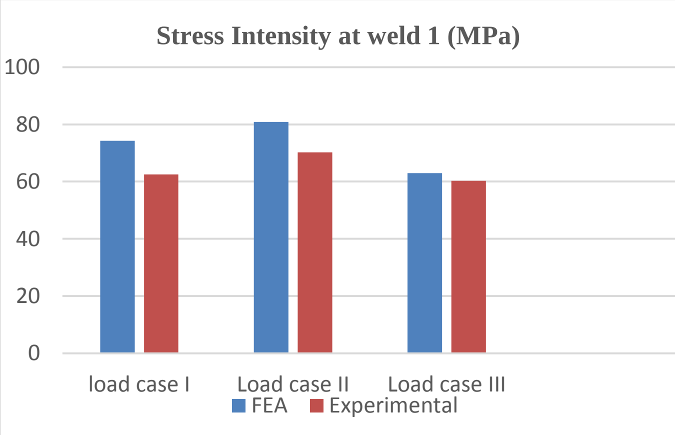
VI. EXPERIMENTATION.

As per the standard criterion given by ASME 2013, edition "Div III and IV" test cycles assumed and virtual testing carried out. The material selected and verified from ASME 2013, edition "Div I and II" section VIII. The entire testing carried out at the specific environment so that no chemical irradiation caused at the low pressure as well as temperature conditions. Due to elevated pressure, sometimes collision occurs. One by one moments and forces with respective direction applied on varispring, it leads into unstable conditions while loading but law of equilibrium grouted by vertical loads. The process time of one load case is assumed to be 22 hours and parallelly others upto 8 hours trial runs. Accordingly we got the values of deflections which changes original place to other for all the X Y Z directions respectively. The deflection values measured by the vernier calliper by applying gauge factors. As from the PLC logic controller plotted the stress cycles, force vs displacement and obtained maximum peak stress values. The entire testing assumed to be connected with PLC it can be easily predict the load cycles with convergence. At the same time pressure value changes with the help of manual operates oried (MOR).

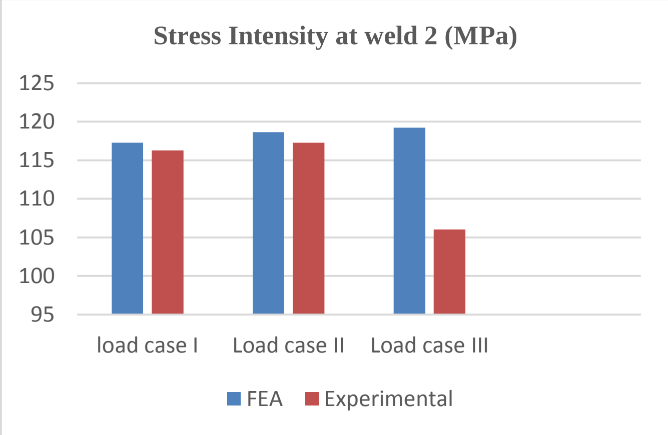
Sr No	Load Case	Deformation (mm)	Stress Intensity	
			Weld1 (MPa)	Weld 2 (MPa)
1	Operating condition	0.3933	62.526	116.285
2	Maximum condition	0.4199	70.265	117.256
3	Shutdown condition	0.3785	60.32	105.32

Table.3 Experimental Results

VII. RESULTS AND DISCUSSION



(a)



(b)

Fig.13. Comparison of FEA and Experimental stress intensities at (a) weld1 (b) weld 2

The maximum stress intensity for weld1, found by FEA as 80.9 MPa and by experimentally 70.2 MPa, in maximum loading condition. A small difference upto 15% in FEA and Experimental results is observed for three load cases, which is negligible.

The maximum stress intensity for weld2, experimental method is 117.2 MPa, in maximum loading condition and from FEA 119.2 MPa in shutdown condition. A small difference is observed of about 12% in results FEA and Experimental method for three load cases, which is acceptable.

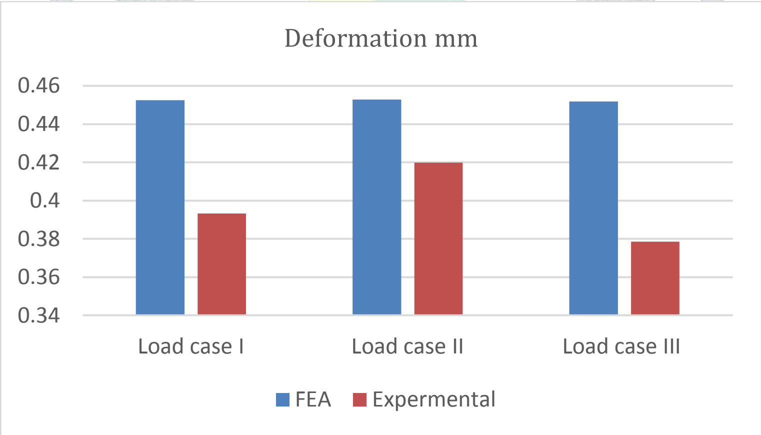


Fig. 14. Comparison of Deformation results

The max deformation of 0.4529 mm and 0.42 mm occur in second load case of maximum loading condition in FEA and experimentally respectively. The difference observed between FEA and experimental result varies upto 18% for three load cases

VIII. CONCLUSION

- As from the observation of three load cases, the Max deformations obtained in maximum loading condition as 0.4529 mm, which is appropriate as per the ASME criterion.
- The Maximum Stress Intensities obtained at Weld I as 80 Mpa and at weld II as 119.2 Mpa in Maximum loading condition, which are within the limit of yield strength of 240 Mpa.
- Comparative study of the results obtained by FEA & experimental method shows that the results are close to each other.
- The material selected and design for component are safe.

.

REFERENCES

[1] ASME Boiler and Pressure Vessels Code Section VIII 2013 Division I & II.

[2] Review Paper on the Faliure Analysis of Weld: Neck Flanges by Dennis Noble Ebenezer G. Alex Akhil Thomas John Vishak G. Richu Thomas Department of Mechanical Engineering, Amal Jyothi College OF Engineering, Kanjirapally

[3] Finite Element Analysis Of Nozzle For Vertical Pressure Vessel by Vijayraj G Parmar, D. S. Shah

[4] Design and Optimization of Weld Neck Flange for Pressure Vessel by Author Vinod Kumar, Vivek Pratap Singh, Surjit Angra, Sushma Rani

[5] Optimization of Spot Weld in Steel Component using FEA Technique by Susheel S. Pote, R.A.Kathar, N.A.Kharche

[6] Design and Analysis of Welding Fixture for Automotive Component using FEA by C. A. Kubade, Dr. S.V. Patil, Mr. V. P. Patil

[7] Investigation Of Residual Stresses and Distortion In Welded Pipe-Flange Joint of Different Classes by Muhammad Abid And Sattar Ullah

[8] Finite Element Analysis and Optimization of Weld Distortion in Automobile Chassis by Akshay Nighot, Prof. Anurag A. Nema , Prof. Anantharma A

[9] Failure Analysis of a High Pressure Butt-Weld Flange by F. Ahmed, F. Hasan and L. Ali

[10] A Textbook of Machine Design by R.S.Khurmi and J.K.Gupta

[11] A Textbook Strength-of-material-by R K Bansal