SSRG International Journal of Mechanical Engineering
ISSN: 2348 – 8360 /doi:10.14445/23488360/IJME-V7I6P107

Volume 7 Issue 6, 41-46, June 2020
© 2020 Seventh Sense Research Group®

# Analysis of Welded Joint Used in Pipeline Support Using Finite Element Method

Samad Gadiwale[1*], S. A. Kore[2]

[1, 2] *Department of Mechanical Engineering, PadmabhooshanVasantraodadaPatil Institute of Technology, Budhgaon, Maharashtra, India-416304*

## *Abstract*

*The design of the welded joint is a critical issue in the offshore industries. This paper presents the studies on the fillet welded joint analysis used in the offshore industry pipeline support. A fillet welded T-joint was designed theoretically according to the specifications. Linear static analysis was carried out in detail using the finite element method (FEM) for finding out the peak stress locations at the root of the joint. The experimental stress analysis of the joint was performed using the strain gauge technique. The results obtained using experimental analysis were in good agreement with the FEM analysis results. The T-joint dynamic response analysis was carried out at the end, using FEM to simulate the real loading conditions.*

## *Keywords*

*Fillet welded T-joint, finite element method, experimental stress analysis, strain gauge, dynamic analysis*

## I. INTRODUCTION

Pipe supports are integral parts of the piping systems in offshore and onshore industries. The pipe support's primary purpose is to transfer the pipe loads to the structure's frame during the operation and sustain the designed conditions. This support also plays a role in carrying the piping system's hydraulic thrust and absorbingits vibration. The ASME codes are generally practiced for the design of the pipe supports. The chemical industries are trending towards lightweight structures in recent years. The piping industry is also not exempted from this. Several pipe supports are used in a single installation plant in offshore industries. A rigorous analysis of the pipe supports can provide the advantage of a material reduction in the system. It also facilitates the peak stress locations in the joints subjected to variable piping loads that can benefit from predicting the support's fatigue life. Trunnion supports are widely used for supporting the pipe structures. These trunnions involve welded and bolted joints. The welded joints in the trunnions are subjected to multiaxial loadings during the operation. Hence, the detailed analysis of such welded joints is evident in many instances during the design process. About this, ample research has been carried out about the stress analysis of the welded joints. Tahami et al. [1] performed the FEM analysis of the fillet welded T-joint used in steam pipes. The effect of pipe thickness and the current input to the heat source were investigated on the burn-through risk. Increased pipe thickness does not allow heat waves from the weld pool to penetrate the pipe and reduce the burn-through risk. Chee and Bakar[2]carried out the dynamic modal analysis of the fillet welded T-joint to predict the structure's natural frequencies and compared with experimental results. The simulation of fusion welding was performed, including moving heat sources in [3]. The thermal analysis followed by mechanical analysis was carried out for the butt-welded joint application—Srivastava, and Raomodeled the residual stresses in the butt-welded joint using FEM. The 2-D thermal-structural coupling was modeled to estimate the residual stresses across the steel plate. The X-ray diffraction method was used for experimental validation [4]. Vivio[5] proposed a theoretical approach to the structural behavior of welded joints. This approach facilitates the accurate estimation of the local elastic stiffness, conveniently used in FEM simulations. The assessment of the fatigue life of the welded joint used in naval structures was performed in [6]. First, the FEM analysis was carried for the structure subjected to constant or variable amplitude loading. Then the two-scale damage model is used to predict fatigue life based on the structure's elastic shakedowns. Pakandam and Farahani[7] investigated the welded joints' fatigue life under uniaxial loading using energy-based fatigue life estimation methods. The welding parameter, such as electric current, heat source, and residual stress distribution, are the critical parameters affecting the welded joints[8–10]. The peak stress method based on the notch intensity factor using FEM was used to estimate the tube steel joints' fatigue life[11]. The analysis of fatigue crack propagation was performed in [12] subjected to bending loads using FEM and followed by experimental validation. It was observed that surface crack was formed with flat semi-elliptic nature and propagated up to 80% before the failure.

In this paper, the fillet welded T-Joint analysis used in trunnion supports for the pipe was carried out using FEM. The effect of load on the peak stress locations was predicted. Experimental stress analysis using the strain gauge technique was performed to validate the FEM results. To understand the behavior of the structure under dynamic loading, initially, FEM based modal analysis was carried out. Then transient analysis was performed to predict the stress variations in the welded joint. Evaluating the stress distribution inside the fillet weld and predicting the peak stress locations and T-Joint is the primary objective of this work.

## II. THEORETICAL DESIGN OF FILLET WELDED T-JOINT

The first step in the welded joint design process is to determine the specifications which can withstand the loading conditions. In the simpler approach, the specifications are selected and then analyzed whether they satisfy the design constraints. Figure 1shows the schematic details of the fillet welded T-Joint used in trunnion supports in piping industries. The 300 [N] or 30 [kg] pipe load was applied, and stress level based on theoretical formula was evaluated. The resultant shear stress of the welded joint subjected to bending load is given by [16]

$$\tau = \left(\frac{\sigma_b}{2}\right)^2 + \tau_p^2 \tag{1}$$

where $\tau$ represents the resultant shear stress, $\sigma_b$ is denotes the bending moment, $\tau_p$ is the primary shear stress. The shear stress value was evaluated using the theoretical equation. The resultant shear stress of 27.80 $[N/mm^2]$ was obtained for the load of 300 [N].

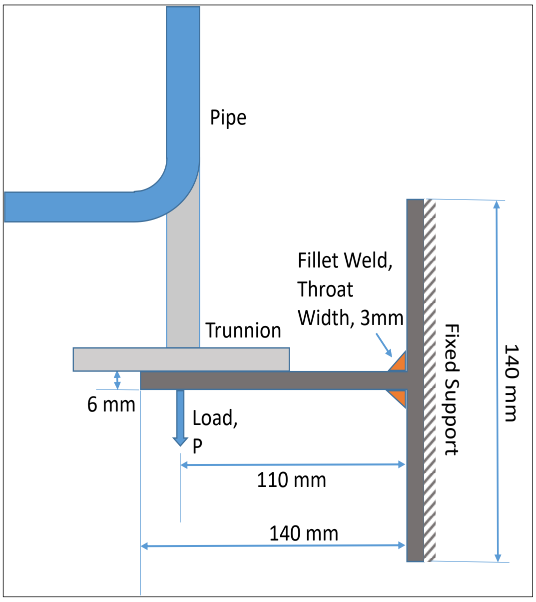

**Figure 1 – Schematic of fillet welded joint with specifications**

## III. ANALYSIS OF FILLET WELDED T-JOINT USING FEM

The simplified geometry of the fillet welded joint used for FEM analysis is shown in Figure 2.The finite element method (FEM) is the popular numerical method commonly used for stress analysis in every industry. Commercial Comsol software was used to solve the FEM model. Geometry was simplified to facilitate the simulation. The four holes were provided for the bolted support in the base plate. The top plate also consists of one hole for clamping the bolt to connect with a structural loading element. These holes in the geometry were not part of real trunnion support in the field but were made to facilitate the experimental testing. However, providing such holes do not alter the loading conditions in the fillet weld region. The tetrahedral mesh elementswere used for the discretization of the structure. The dense mesh was used in the fillet weld region to increase accuracy. The size and the number of elements were selected according to the convergence requirement. The fixed support boundary condition was applied at the holes in the base plate. The load analogous to the actual loading condition was applied at the hole in the T-plate. The material properties used in FEM simulation are summarized in Table 1.

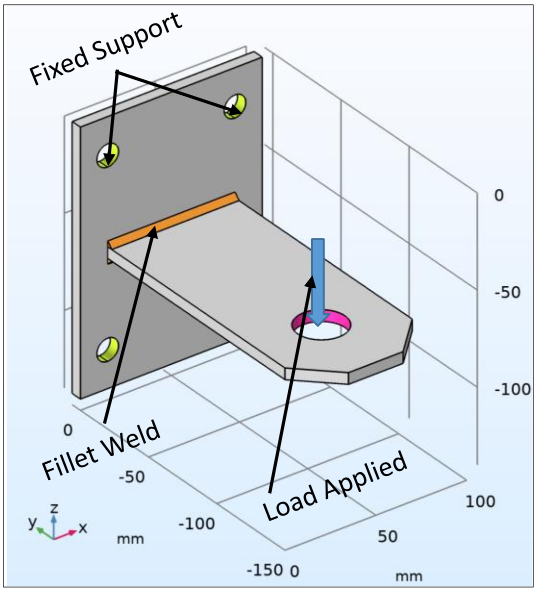

**Figure 2 – Simplified geometry of fillet welded T-Joint used in FEM analysis**

Initially, the static analysis was performed to evaluate the stress distribution and the structure's peak stress locations. The equations used for solving the static analysis problem can be written as [14]

$$\nabla \cdot (FS)^T + Fu = 0 \tag{2}$$

Where $F$ is the deformation gradient tensor, $S$ is the stress tensor, and $u$ is the displacement vector. The load of 300 [N] was applied, and the stress level was evaluated. The effect of the load on the maximum principal stress and shear stress was estimated.

**Table 1 – Material properties used in FEM analysis**

| | Structural Steel (ASTM A36) | Weld Filler Material (E100) |
|---|---|---|
| Modulus of Elasticity [N/mm$^2$] | 2.00e5 | 2.10e5 |
| Modulus of Rigidity [N/mm$^2$] | 0.78e5 | 0.72e5 |
| Poisson's Ratio | 0.26 | 0.38 |
| Density [Kg/m$^3$] | 7800 | 7139 |
| Yield Strength [N/mm$^2$] | 2.50e5 | 3.93e5 |
| Tensile Strength [N/mm$^2$] | 4.50e5 | 4.82e5 |

To know the natural frequency and the response of the structure when subjected to transient loading. The equation solved for modal and transient analysis is written as following[14]

For Modal Analysis,

$$\rho\omega^2 u + \nabla\cdot S, \qquad -i\omega = \lambda \tag{3}$$

For Transient Analysis,

$$\rho\frac{\partial^2 u}{\partial t^2} = \nabla\cdot S + Fv$$

Where $\rho$ is the density of the material, $\omega$ is the natural frequency of the structure, $Fv$ is the applied volume force. The sinusoidal load with 100 [Hz] frequency and 100 [N] amplitude was applied in the y-direction, whereas a sine load of 700 [N] with 200 Hz frequency was applied in the z-direction. The nature of the applied load is shown in figure 3. The response of the structure was analyzed for 50 milliseconds.

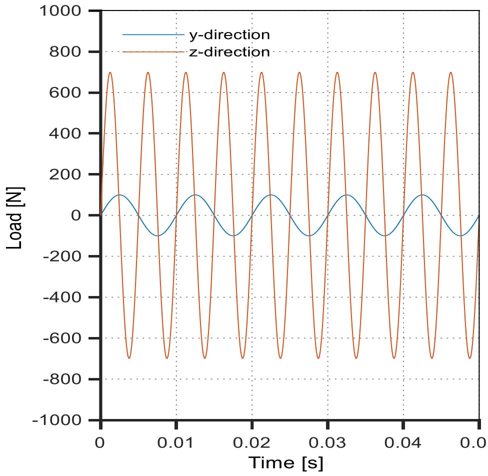

**Figure 3 – Transient load applied to T-Join in FEM Analysis**

## IV. EXPERIMENTAL STRESS ANALYSIS

Figure 4 shows the schematic of the experimental stress analysis. The strain gauge technique was used to determine the stresses in the structure subjected to static analysis. A power screw frame was designed and manufactured to apply the gradual load on the T-Joint. The fillet welded T-Joint was mounted on the fixture. The fixture was supported on the base plate. The load cell was mounted below the base plate such that load transferred from the power screw can be directly transferred and measured using a load cell. The 0-45-90 type strain rosette, UFRA-1-350 (Tokyo Sokki Kenkyujo, TML), was firmly mounted on the fillet weld using epoxy resin. The load was gradually applied from 300 [N] to 900 [N] on the T-Joint, and strain developed in the strain rosette was measured using a custom made strain gauge circuit. The MATLAB software was used for further computation of the stresses. Figure 5 shows the actual experimental setup. The data obtained using strain rosette is shown in Table 2.

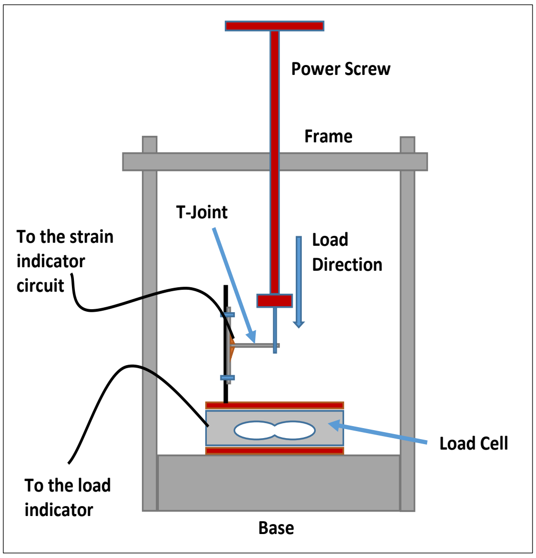

**Figure 4 – Schematic of the experimental setup**

**Table 2 - Strain Rosette readings**

| Sr. No | Load [N] | Strain Rosette Data [In micro-strain] | | |
|---|---|---|---|---|
| | | $\varepsilon_a$ | $\varepsilon_b$ | $\varepsilon_c$ |
| 1 | 300 | 0 | 0.00054 | 0.00045 |
| 2 | 400 | 0 | 0.00058 | 0.00061 |
| 3 | 500 | 0 | 0.00065 | 0.00078 |
| 4 | 600 | 0 | 0.00077 | 0.00095 |
| 5 | 700 | 0 | 0.00091 | 0.00108 |
| 6 | 800 | 0 | 0.00112 | 0.00119 |
| 7 | 900 | 0 | 0.0012 | 0.00133 |

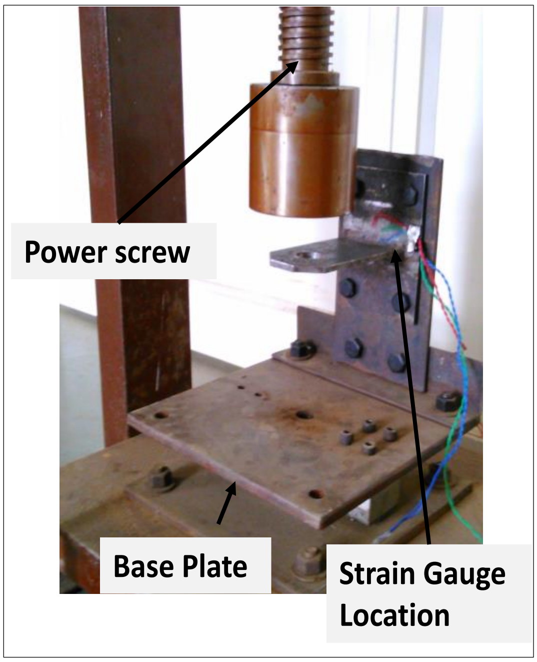

**Figure 5 – Actual experimental setup**

The equation used for the principal stress calculation are described as[15]

$$\sigma_{1,2} = \frac{E}{2}\left[\frac{\varepsilon_a+\varepsilon_c}{1-\nu} \pm \frac{\sqrt{2}}{1+\nu}\sqrt{((\varepsilon_a-\varepsilon_c)^2+(\varepsilon_b-\varepsilon_c)^2)}\right] \qquad \textbf{(4)}$$

where, $\sigma_{1,2}$ represents the principal stress, $E$ is the modulus of elasticity, $\nu$ is the Poisson's ratio, and $\varepsilon_a, \varepsilon_b, \varepsilon_c$ are the strain measured from 0-45-90 strain rosette?

## V. RESULTS AND DISCUSSION

In this section, the discussion of the results was carried out. The effect of the load on the stress distribution was discussed in detail. At first, static analysis results were compared, and then the results of modal analysis and transient analysis were presented.

Figure 6 shows the stress distribution at the fillet weld locations for the 300 [N]. The surface plot clearly shows the localization of stress levels at the fillet weld. The shear stress value of 29.80 [N/mm$^2$]was estimated, which was in good agreement with the theoretical value.

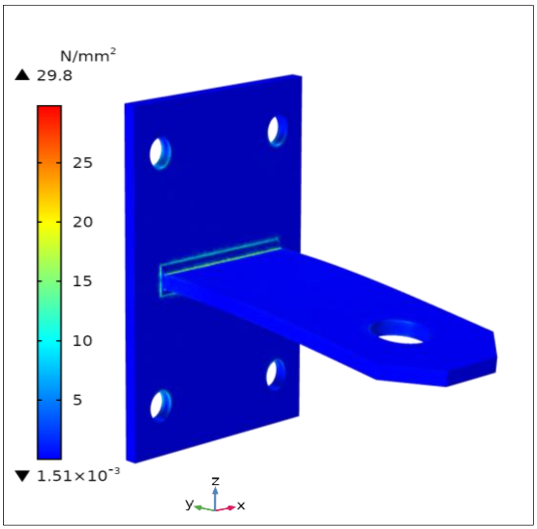

**Figure 6 – Shear Stress distribution in T-Joint at 300[N] loading**

The permissible value is 145 [N/mm$^2$] for shear stress and 290 [N/mm$^2$] for bending stress.Figure 7 shows the contour plot of maximum principal stress in the structure. Both the shear stress and principal stress values are well below the allowable stress level in shear and bending. Figure 8 shows the effect of load on the shear stress and principal stress on the fillet weld joint. The shear stress level increased from 25.90 to 77.60 [N/mm$^2$] for the FEM analysis as the load increased from 300[N] to 900[N] gradually. The values are well below the permissible shear stress level for the given material.

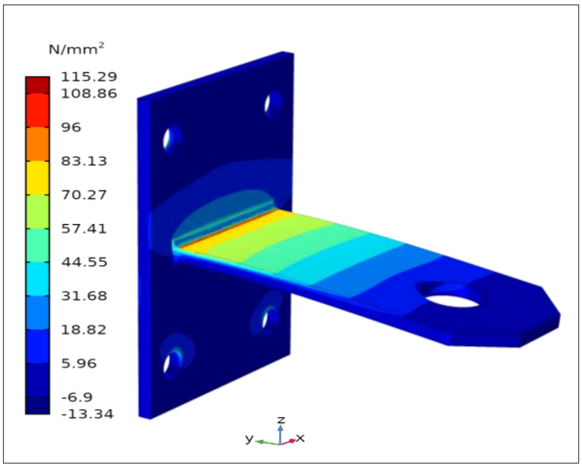

**Figure 7 – Principal stress distribution of T-Joint at 300[N] loading**

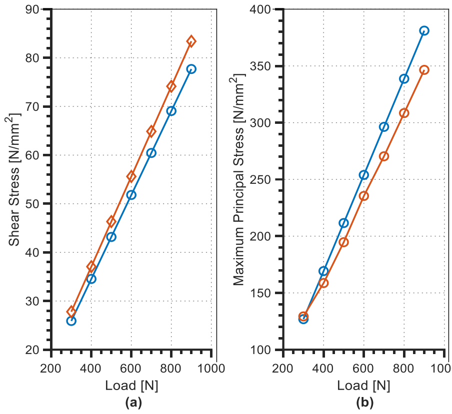

**Figure 8 – Effect of load on (a) Shear stress (b) Principal Stress**

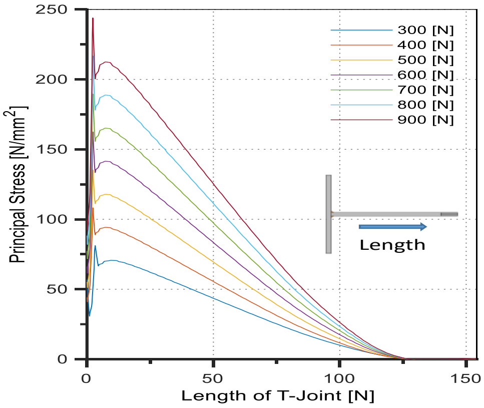

**Figure 9 – Principal stress distribution across the length of T-Joint**

The shear values obtained from FEM analysis were compared with the theoretically calculated values. The FEM values were in close agreement with theoretical values. Similarly, the principal stress values were also obtained from FEM analysis and compared with experimental values obtained from strain gauge readings. It was observed that the principal stress values increased beyond the permissible bending stress values when the load was increased from 600 to 900 [N] gradually. Besides, the principal stress distribution across the length of the T-Joint is shown in Figure 9. The stress level is very high at the root of the fillet weld and exponentially decreases across the length. The modal analysis was also performed using FEM on the same model to know the structure's natural frequency. The first natural frequency was observed at 237.71[Hz], and the second mode was observed at 796.44[Hz]. The first two modes were the flexural type. The third mode was observed at 1069.9[Hz], which was the torsional type. The estimated natural frequencies are higher enough to avoid low-frequency vibration.

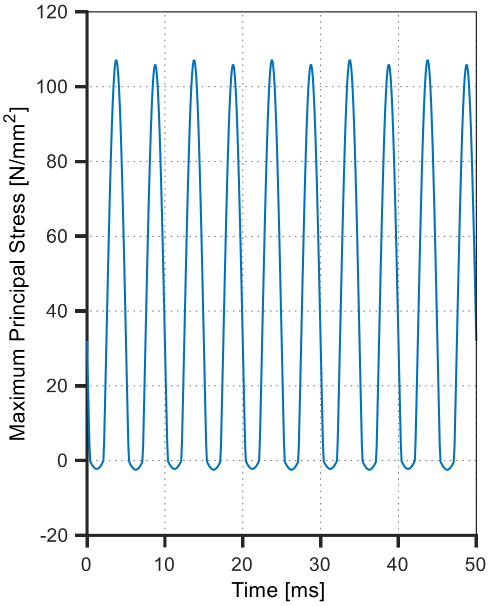

**Figure 10 – Stress variation of the T-Joint subjected to transient loading**

In the end, a transientanalysis was performed. The Rayleigh damping was considered for the analysis. The value ofthe mass damping coefficient and stiffness damping coefficient was considered 50[1/s] and 1e-4[s]. The combined sinusoidal loading was applied at the T-Joint, as shown inFigure 3. The analysis was carried out for 25[milliseconds] to resemble the impact of the pipe load. The averaged principal stress level at the fillet weld was evaluated. The stress variation of the fillet weld over time is shown in Figure 10. The stress also varied in a sinusoidal manner, similar to the load applied. The maximum stress level was reached 110[N/mm$^2$],below the material's allowable bending stress.

## VI. CONCLUSION

The rigorous analysis of the fillet welded T-Joint was performed at work. Such joints are commonly used in the trunnion supports, which are used in piping systems. Initially, the static analysis using FEM was performed. The effect of load on the shear stress level and the main stress level were compared. It was observed that shear stress values obtained from the FEM analysis were in the permissible limit. However, the principal stress increased beyond the permissible bending stress level when the load increased above 600[N]. The FEM analysis values were in good agreement with the experimental values. However, no visible failure was observed when the load increased above 600[N] while testing. In the end, modal analysis and transient response analysis of the T-joint were performed to understand the dynamic behavior. The estimated natural frequencies were high enough to avoid the low-frequency vibrations of the system. The transient load similar to actual loading in operation was applied, and the fillet weld's stress variation was observed. The stress levels were found to be within permissible limits for applied loading. Such fillet welded joints are generally subjected to fatigue loading due to the pressure variations in the pipe. In addition, wind loading, earthquakes also contribute to the multiaxial loading situations of the fillet weld. The fatigue analysis of such welded joints is a challenging task and aimed and future work.

## CONFLICT OF INTEREST

The authors do not have any conflict of interest regarding this work.

## REFERENCES

[1] Vakili-Tahami F, Zehsaz M, Saeimi-Sadigh MA and Seyedreyhani, Finite element analysis of the in-service-welding of t joint pipe connections, Eur. J. Sci. (2010) Res.40 557–68.

[2] Chee NC, Rahim A and Bakar A ,Finite element modeling of arc welded joints, J. Mek. (2010) 15–30.

[3] Anca,Andres,Cardona,Alberto,FachinottiV, Finiteelementmodeling of welded joints,Asoc.Argentina Mecánica Comput.XXVII, (2008 ) 10–3.

[4] Srivastava V and Rao A Modeling of Residual Stresses in a Butt-welded Joint with Experimental Validation Proceedings oftheCOMSOL Conference Pune, India .(2015).

[5] Vivio F, A new theoretical approach for structural modeling of riveted and spot welded multi-spot structures,Int. J. Solids Struct. (2009) 46 4006–24.

[6] Erny C, Thevenet D, Cognard J-Y and Korner M, Fatigue assessment of naval welded assemblies,Procedia, (2010)Eng.2 603–12.

[7] Pakandam F and Varvani-FarahaniA, A comparative study on fatigue damage assessment of welded joints under uniaxial loading based on energymethods Procedia ,(2010) Eng.2 2027–35.

[8] Qu W, Wang M and Zhou Q , Effect of Welding Residual Stress on Along-Wind Fatigue Life of Welded Joints in Guyed Mast, Shock Vib.(2020).

[9] Khoshroyan A and Darvazi A , Effects of welding parameters and welding sequence on residual stress and distortion in Al6061-T6 aluminum alloy for T-shaped welded joint Tran"s. Nonferrous Met. Soc. China, (2010) English Ed.30 76–89.

[10] Barsoum Z and Barsoum I, Residual stress effects on fatigue life of welded structures using LEFM, (2009)Eng. Fail. Anal.16 449–67.

[11] Campagnolo A, Vormwald M, Shams E and MeneghettiG,Multiaxial fatigue assessment of tube-tube steel joints with weld end using the peak stress method,Int.J. Fatigue135 105495 (2020).

[12] Baik B, Yamada K, and IshikawaT,Fatiguecrack propagation analysis for welded joint subjected to bending, Int. J. Fatigue33 746–58 (2011).

[13] Neha B. Thakare, A. B. Dhumne, A Review on Design and Analysis of Adhesive Bonded Joint by Finite Element Analysis, SSRG International Journal of Mechanical Engineering. 2(4) (2015):17-20.

[14] COMSOL,Structural Mechanics Module User's Guide (Version 5.4), (2018).

[15] VISHAY PRECISION GROUP Strain Gage Rosettes: Selection, Application and Data Reduction(2010).

[16] Bhandari V B, Design of Machine Elements(Tata McGraw-Hill Education) (2010).