Analysis of Fillet Welded Joint for Supporting Structure in Pipe Industries

Mr. Mulla Azhar S[1] Mr. Kulkarni P.D[2]
[1]Student [2]Associate Professor
[1,2]Department of Mechanical Engineering
[1,2]SPPU Pune, SIT Lonavala, India

Abstract— In the recent years, the welded joints becomes an integral part of assemblies used in the industries. In fact in piping industries welded joints are crucial and prone to different types of loadings. Hence, in such applications, most of the failures are due to the failing of the welded joints. Hence, a modern computational approach based on finite element analysis for strength assessment has been used in this dissertation work. Linear finite element analysis has been carried out to determine the peak stresses which is followed by the modal analysis to decide the mode of vibration. The results of FEM analysis have been validated experimentally using strain gauge technique.
Key words: experimental, FEA, fillet welded joints, modal analysis, offshore industries, supporting structure

I. INTRODUCTION

Welding is defined by the American Welding Society (AWS) as a localized union of metals or non-metals produced by either heating of the materials to a suitable temperature with or without the application of pressure, or by the application of pressure alone, with or without the use of filler metal.

Welding techniques are one of the most important and most often used methods for joining pieces in industry. Welded joints used in almost every industries depends on various applications and where the permanent joints with high strength are necessary. Some of the applications, where welded joints used are the structural supports, automotive joints, piping industries, pressure vessels, etc. The latest trends in the industries are focusing on the high strength; high rigidity welded joints for different metals with the advancement in the welding technology. Any information about the shape, size and residual stress of a welded piece is of particular interest to improve quality.

Welded joints used in the industries subjected to various types of loading such as axial, bending or torsional. Sometimes these loads are dynamic in nature. Most of the time these loads act together. Hence, the rigorous analysis of welded joint is very complicated in such cases involving a detailed study of both the rigidity of the parts joined and the geometry of the weld. The various incremental segments of the weld serve as a multiplicity of redundant attachments, each Carrying a portion of the load dependent on its stiffness. The well-known procedures of welded joint analysis are based on Simplifying assumptions commonly used to obtain results sufficiently accurate for engineering use in most applications.

Hence, it is necessary to carry out accurate stress analysis of welded joints to overcome the errors occurred in the strength estimation using conventional methods. Therefore, in this dissertation work the structural analysis of fillet type welded joint used in structural supports for pipes in offshore industries has been carried out using finite element method (FEM).

From the review of the literature on welded joints, it is seen that there is some scope for research work in the area of structural analysis of welded joints. As such, it is proposed to carry out some theoretical and experimental studies on structural analysis of fillet type welded joint used in structural supports for pipes in offshore industries using finite element method (FEM). For this purpose the following work will be carried out.

II. THEORETICAL ANALYSIS

A cantilever beam of rectangular cross –section is welded to the support using two weld $W1$ and $W2$ as shown in fig. According to the principle of Applied Mechanics, the eccentric force P can be replaced by an equal and similarly directed force P acting on the plane of welds

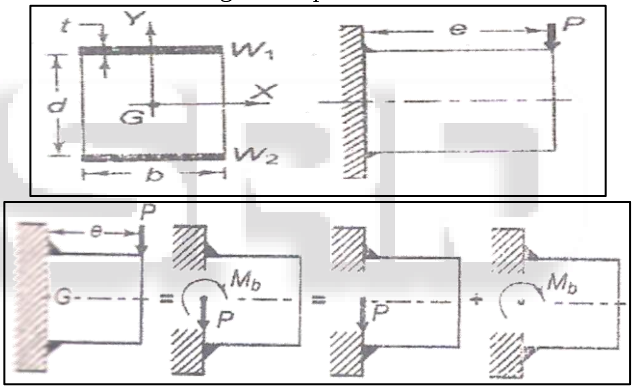

Fig. 1:

By considering design of two symmetrically welded joints we can calculate the shear stress for different loads by using formula

$$\tau = \sqrt{(\frac{\sigma_b}{2})^2 + (\tau_1)^2}$$

III. STRESS ANALYSIS BY USING FEA

1) Pre-Processing-This is the vital step in the FEM analysis. It includes the CAD modeling, Meshing and applying the boundary conditions. This involves the element selection, assigning material properties and real constants.
2) Solution-It involves the actual solving of the differential equations for each element and finding out the unknowns. This job was done by the software itself in the computer. However, the user has to select the method of the solution based on the type of the problem.
3) Post-Processing-This step includes the viewing of the results, interpretation of results and verification of the data obtained and conclusions.

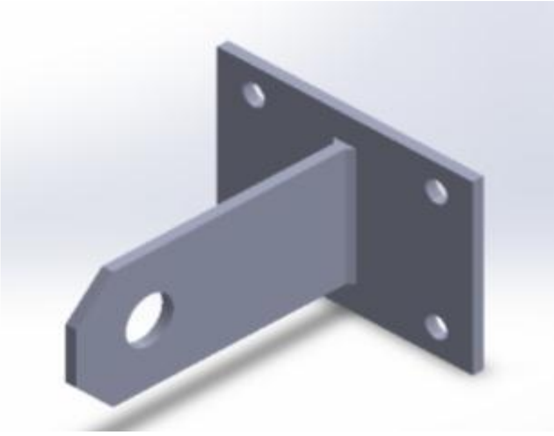

Fig. 2: CAD model prepared for analysis

Material Properties	Value
Modulus of Elasticity	2.07E5 N/mm^2
Poisson's Ratio	0.34
Density	7.139E-9 $Tonns/mm^3$

Table 1: Material Properties of Structural Steel

Material Properties	Value
Modulus of Elasticity	2.01E5 N/mm^2
Poisson's Ratio	0.38
Density	7.139E-9 $Tonns/mm^3$

Table 2: Material Properties of Weld

By using above properties as input and applying boundary conditions, FEA analysis is carried out. The results of the analysis are tabulated and compared in the table below.

Welded Joint Specimen	*Shear Stress*		
Load	Theoretical	FEA	% Error
N	*N/mm²*		
343.35	15	17.21	12.841
686.7	32.64	36.41	10.354
1030.05	50.97	60.04	15.106
1373.4	70.64	78.87	10.434

Table 3: Shear Stress Comparison for The Therotical and Fea Analysis

IV. MODAL ANALYSIS OF WELDED JOINT

Modal analysis is used to determine the natural frequencies and the mode shapes of the components/systems. The natural frequencies and the mode shapes are important parameters in the design of a structure for dynamic loading conditions and are a starting for more detailed analysis such as harmonic response analysis, transient analysis, etc.

The modal analysis of a welded joint was carried out to find its fundamental frequency. Modal analysis is carried out at the free-free condition. To perform the modal analysis the same ANSYS 13.0 software was used with some modifications in the material properties. In this case, it is necessary to define only young's modulus of elasticity and the density of the material. The density of the steel material and weld fill material is assigned to the model. The mapped meshing is consider for modal analysis to reduce the number of elements. SOLID1855 is the element used for mapped meshing. The mode extraction is done by different methods in ANSYS 13.0 software such as Block Lanczos, Subspace, and Power Dynamics, etc. The Block Lanczos method is the default method and recommended for most of the applications. This method can extract more than forty mode shapes. It is typically used in complex model as it handles the different types of elements at a time and also the rigid body models. The mode are normalized to mass matrix option in ANSYS 13.0 by default which allows using the modes in further mode superposition analysis such as transient analysis or spectrum analysis. Such normalized mode can be used in FEM based fatigue analysis.

Mode Number (n)	Natural Frequency (Hz)
1	426.19
2	1550.0
3	1823.3
4	2415.5
5	3611.7

Table 4:

In the free-free condition modal analysis first, five natural frequencies are observed significantly in FEM analysis. Hence, the first natural frequency is considered as the fundamental frequency. It is 426.19 Hz by FEM which is quite higher.

V. EXPERIMENTAL STRESS ANALYSIS

The method for the experimental stress analysis of the welded joint based on the electric resistance strain gauge technique is presented. This involve the selection of strain gauge or rosette, bonding material, strain gauge circuits, and use of dummy gauge and the experimental methodology. The experimental setup was developed to carry out the stress analysis.

A. Strain Rosette

When a state of strain at a point and the direction of the principal stress or strain is known, then the single gauge can be oriented along this direction and strain measurement is carried out.

However when the state of strain is not known, then three or more strain gauges may be used at the point under consideration to determine the strain at that point. The resulting configuration is known as strain rosette. The figure shows the typical three gauges rectangular strain rosette having orientation 0-45-90 degrees with each other. It means that gauge one is horizontal and other two gauges at 45 degrees and 90 degrees with gauge one respectively. In this way, strain in three different directions is measured.

B. Experimental Set Up and Strain Measurement

In this experimentation, the directions of principal stresses were unknown, but the location is finding by earlier FEM analysis results. The gauge location is then fixed. Therefore, linear strain gauge rosette is found suitable and hence was selected for the purpose. Finally, the gauge continuity and installation resistance were checked by the multi-meter. The quarter bridge configuration is used in Wheatstone bridge along with dummy gauge arrangement for temperature compensation.

The experimental setup was used as shown in Figure for applying the static load with the help of power screw arrangement

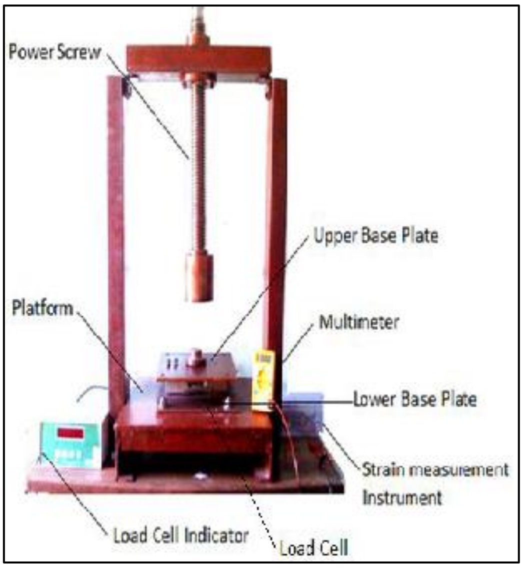

Fig. 3: Experimental set up

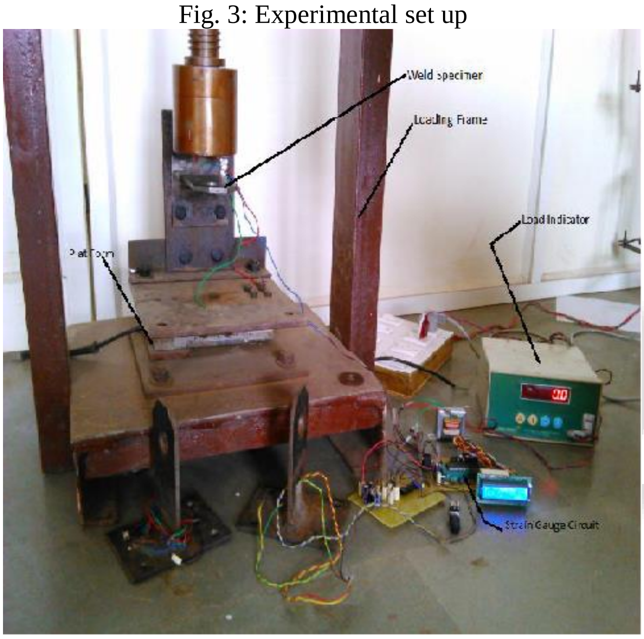

Fig. 4: Welded Joint Specimen Mounted on Experimental Setup

The welded joint is hold between the upper base plate and power screw cup. There was a thrust bearing provided between the power screw and cup to provide free rotational movement of the cup while, the screw is gradually moving up and down. Below the upper base plate the load cell was mounted to measure the load coming on the welded joint in Newton. The lower base plate is rested and fastened to a platform. To guide the power screw movement, two vertical beams are provided and one horizontal beam which guides the power screw arrangement and resting on these vertical beams. The setup is easy to disassemble.

The load is gradually applied in the spring and the output voltage difference of the quarter bridge Wheatstone circuit is measured on multi-meter regarding microvolts. Readings are taken for different loads, and necessary calculations were done in Microsoft Excel to get the required strain readings. To avoid strain hardening effect, the strain measurements were done at loading as well as unloading. The table shows the values in microstrain for different loads.

Strain Readings for Loading and Unloading

Load	Epsilon A	Epsilon B	Epsilon C
	Microstrain		
343.35	0	0.00044	0.00038
686.7	0	0.00083	0.00071
1030.05	0	0.00132	0.00108
1373.4	0	0.001798	0.00134

Table 5:

By using above readings calculate maximum principle stress induced into the specimen.

Max. Principal Stress(N/mm^2)		Error
Experimental	FEM	%
98.02033376	97.56	0.4
184.3102039	195.129	5.54
288.7581561	292.636	1.32
381.6360961	390.257	2.20

Table 6:

By using above reading plot the graph showing the comparison between FEA and experimental values of principle stress.

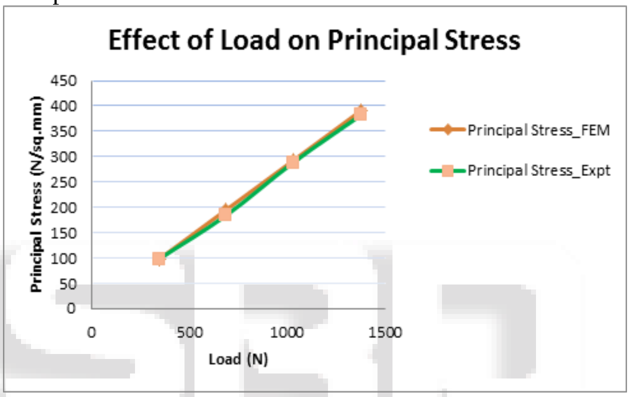

Fig. 5: Load v/s Stress

VI. Conclusion

The peak stress value of welded joints obtained by finite element analysis is in good agreement with that obtained using experimental analysis. This suggests that FEM can used at the design stage of the welded joints.

As the load increases, the stress and deflection values increase. This follow the good linear relationship. But the stress distribution pattern for the welded joints was similar in all loading cases.

As for the given welded joints, the fundamental natural frequency is very high, this structure can used in the pipe supports where dynamic loads are coming. There are very fewer possibilities where resonance can occur.

The welded joints have the effect of residual stress on its strength. So there may be the variations in the theoretical stress values and the experimental stress values. Strain gauge rosette technique can be used for the stress analysis of welded joints and gives results in good agreement.

References

[1] Vivio Francesco, “A new theoretical approach for structural modeling of riveted and spot welded multisport structure” International journal of solids and structures, vol46,2009 pp4006-4024

[2] Al-Samhan A. M., Darwish S. M. H., "Finite Element Modeling Of Weld-Bonded Joints", Saudi Engineering Conference, KFUPM, Dhahran, Vol. 5, 2002, pp 213-225
[3] Carole Erny, David Thevenet, Jean-Yves Cognard, Manuel Körner, "Fatigue assessment of naval welded assemblies", Procedia Engineering, vol. 2, 2010 pp 603–612
[4] Anca Andrés, Cardona Alberto and Fachinotti Víctor, "Finite Element Modeling of Welded Joints", Mecánica Computacional, vol. XXVII, pp 1445-1470
[5] Gucci sky R. V., Petrov S. V., "Fatigue of Fillet Welded Joints Assessment by the FEA Simulation of Damage Accumulation ", Magazine of Civil Engineering, Vol. 4, 2011 pp 5-9
[6] Farid Vakili Tahami, Mohammad Zetas, Mohammad-Ali Saeima-Sadigh, Seyedreyhani, "Finite Element Analysis of the In-service-Welding of T-Joints Pipe Connections", European Journal of Scientific Research, Vol.40 No.4, 2010, pp 557-568
[7] New Chang Chee, Abd Rahim Abu Bakar, "Finite Element Modeling Of Arc Welded Joints", Jurnal Mekanikal, No. 23, 2007, pp 15 – 30
[8] F. Pakandam, A. Varvani-Farahani, "A comparative study on fatigue damage assessment of welded joints under uniaxial loading based on energy methods", Procedia Engineering, vol. 2, 2010, pp 2027–2035
[9] Wahl A.M., "Mechanical Springs", second edition, McGraw-Hill Co. 1963
[10] Shigley J.E., Mischke R.C., Budynas R.G. and Nisbett K. J., "Shigely's Mechanical Engineering Design", Eighth Edition, Tata McGraw-Hill Co., 2009
[11] Rao Singiresu.S., "The Finite Element Method in Engineering", Fourth edition, Elsevier Publications, 2004
[12] Gokhale Nitin, Deshpande Sanjay, Bedekar Sanjeev and Thite Anand, "Practical Finite Element Analysis", Finite to Infinite, 2008
[13] Singh Sadhu, "Experimental Stress Analysis", Fourth Edition, Khanna Publications, 2004
[14] TML Pam, E-101S, Strain Gauge Catalogue, Tokyo Sokki Kenkyujo Co., Ltd
[15] HBM, Strain Gauge Catalogue, Hottinger Baldwin Messtechnik GmbH, Germany
[16] Hoffman K., "An Introduction to Measurement Using Strain Gauge", Hottinger Baldwin Messtechnik Publication, 1989.