International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 07 | July - 2022 Impact Factor: 7.185 ISSN: 2582-3930

FEA Analysis of Weld Joint using ANSYS Software

Vinod Dewangan, Dinesh Dubey***
** Research scholar, Department of Mechanical Engineering, RSR Rungta College of Engineering & Technology, Bhilai .*
Email: dewvinod25@gmail.com
***Assistant Professor, Department of Mechanical Engineering, RSR Rungta College of Engineering & Technology, Bhilai.*
Email: dineshdubey1208@gmail.com

Abstract: The welding process involves joining of 2 or more parts using heat combined with pressure. The objective of current research is to investigate the strength of tube flange joint using techniques of Finite Element Analysis. The CAD designing and FEA simulation is to be conducted in ANSYS simulation package. The most critical regions are observed at the corners of the weld joint. The effect of weld angle is also evaluated. The deformation is observed to be reduced with increase in weld angle linearly. The normal stress is observed to increase linearly and reach to peak value and then decrease linearly to reach minimum value at 140^0 weld angle.

Key Words: *FEA, Weld joint, Optimization*

1. INTRODUCTION:

The welding process involves joining of 2 or more parts using heat combined with pressure. The coalescence is achieved by adding filler material. The joined assembled parts are called weldment. The weld joints have to be properly dimensioned due to high induced stresses.

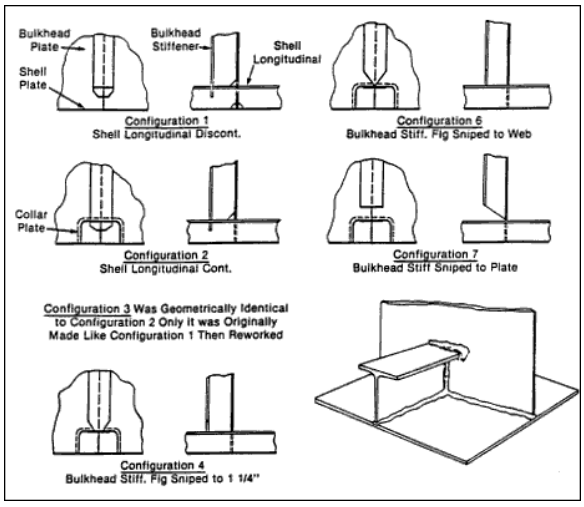

Figure 1: Different types of weld configurations

Different configurations of weld joints are shown in figure 1 above. The weld joints are joined with different cross sections as shown in figure 1.

2. LITERATURE REVIEW

T. Ninh Nguyen and M. A. Wahab[1] have conducted research on weld joint. The author investigates the nature and type of distortions incurred by weld joints. The author concluded that 2 type of loadings i.e,. axial and bending tends to cause the distortions.

Kyungwoo Lee[2] have conducted numerical investigation on weld joint under combined loading conditions. The weld joint is made of Ludwick type material and runge-kutta method is used to investigate the strength of weld joint. The uniformly distributed load is applied on the weld joint and from the research closed form solution is not obtained.

According to Robb C Wilcox [3] have conducted research on strength of filleted weld using analytical techniques. The analysis involved applying loads along transverse and longitudinal directions both. The research findings have shown higher strength for transverse loaded welds as compared to longitudinally loaded welds.

Mahapatra et al. [4] have conducted research on one-sided fillet weld joints using experimental techniques. The effect of single fillet on induced angular distortion is investigated. The research findings have shown that proper application of constraints at specific positions could resist welding distortion.

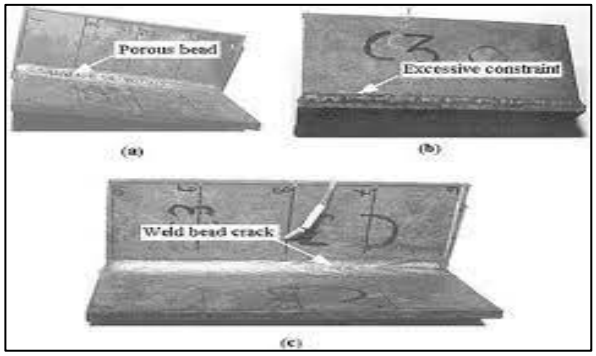

Figure 2: Weld defects on T shaped joint [4]

Kumose et al. [5] have conducted research on pre-straining process to mitigate the angular distortion. Pre-straining involves either "plastic or elastic straining in the direction opposite to distortion before welding is done. Kumose found that the magnitude of plastic pre-strain to avoid distortion was comparatively smaller than that of free angular distortion when the flange thickness is comparatively greater than the weld leg length. Free angular distortion in this research is referring to angular distortion that is free from external forces and only affected by the experimental parameters" [5].

3. OBJECTIVE

The objective of current research is to investigate the strength of tube flange joint using techniques of Finite Element Analysis. The CAD designing and FEA simulation is to be conducted in ANSYS simulation package.

4. METHODOLOGY

The design of tube flange weld joint is developed in design modeler of ANSYS software. The tube flange weld joint is shown in figure 3. The model is developed using sketch and revolve tool.

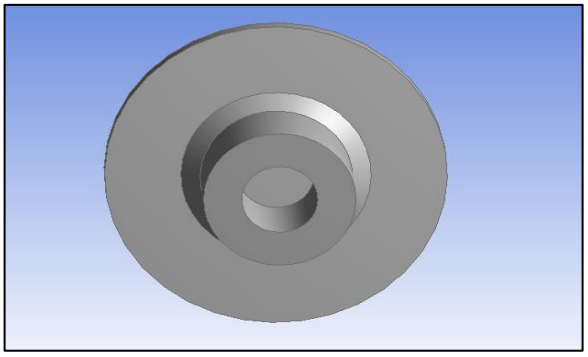

Figure 3: Tube flange weld joint

The model of tube flange weld joint is discretized using tetrahedral element type. The model is meshed with fine sizing and high relevance. The meshing is done with normal inflation. The mesh obtained is of uniform density and uniform distribution as shown in figure 4.

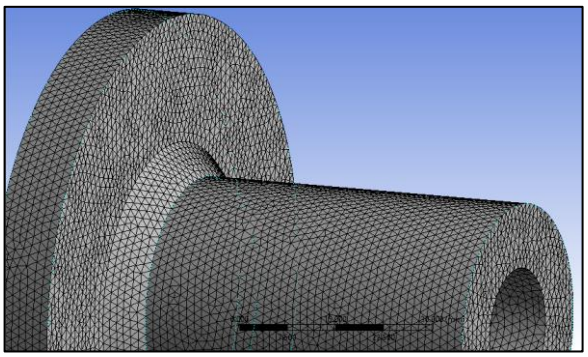

Figure 4: Meshed model of Tube flange weld joint

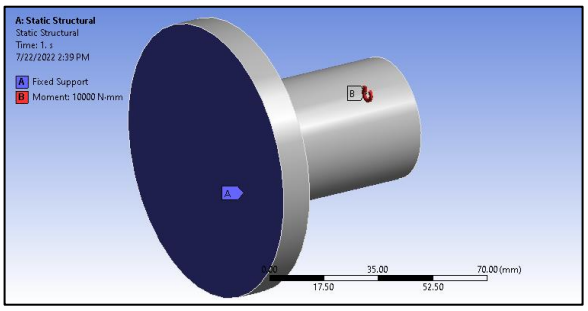

Figure 5: Loads and boundary conditions

The loads and boundary conditions are applied on tube flange weld joint as shown in figure 5 above. The circular face of cylinder is applied with fixed support and moment is applied on other surface with magnitude of 10N-m. The FEA

simulation is run to determine shear stress, deformation and normal stress.

5. RESULTS AND DISCUSSION

From the FEA analysis, the critical regions of high stresses and deformation are obtained. The maximum shear stress is observed at the corner region tube wherein the magnitude is more than 1.2MPa. The shear stress at other regions is more than .75MPa as shown in dark orange colored region.

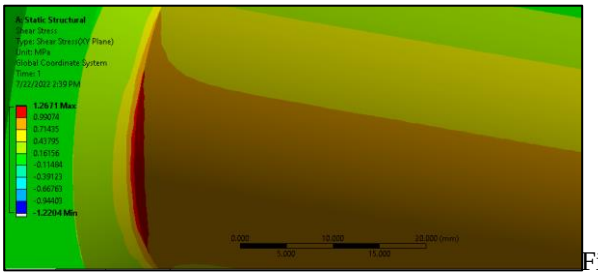

Figure 6: Shear stress plot on weld joint

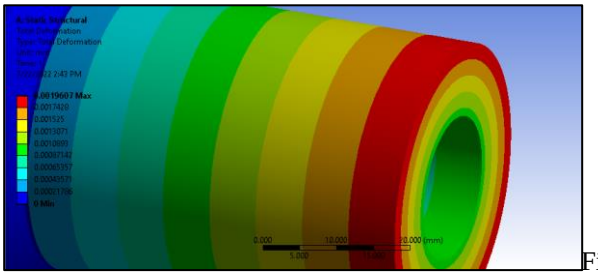

Figure 6: Total deformation plot on weld joint

The total deformation plot is obtained for weld joint as shown in figure 6 above. The maximum deformation is obtained at the free end of weld joint wherein the maximum deformation is more than .001089mm. The deformation reduces on moving towards the fixed support region of weld joint.

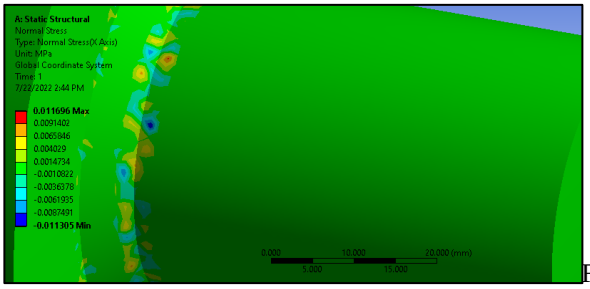

Figure 7: Normal stress plot on weld joint

The normal stress plot is obtained for weld joint. For most of the regions the normal stresses are uniform in magnitude as shown in dark blue colored region. The magnitude of normal stresses at this zone is .00147mm.

5. CONCLUSION

The FEA is a viable tool in determining strength of weld joint. From the FEA analysis the critical regions of high stresses induced on tube flange weld joint is evaluated. The most critical regions are observed at the corners of the weld joint. The effect of weld angle is also evaluated. The deformation is observed to be reduced with increase in weld angle linearly. The normal stress is observed to increase linearly and reach to peak value and then decrease linearly to reach minimum value at 140^0 weld angle.

REFERENCES

[1] T. Ninh Nguyen and M. A. Wahab, “The effect of weld geometry and residual stresses on the fatigue of welded joint under combine

[2] Kyungwoo Lee, “ Large deflections of cantilever beams of nonlinear elastic material under a combined loading,” International Journal of Non-Linear Mechanics 37 (2002) 439–443.

[3] Robb C. Wilcox, “The effect of weld penetration on tensile strength of fillet welded joints”,B.S., Naval Architecture and Marine engineering, U.S. coast guard acadamy, 1991

[4] Mahapatra, M., G. L. Datta, B. Pradhan, and N. R. Mandal. “Modelling the effects of constraints and single axis welding process parameters on angular distortions in one-side fillet welds.” Proc. IMechE 221 Part B: 397-407.

[5] Kumose, T., T. Yoshida, T. Abbe, and H. Onoue. “Prediction of Angular Distortion Caused by One-Pass Fillet Welding.” Welding Journal. 1954: 945-956