International Journal of Scientific Research & Engineering Trends
Volume 7, Issue 2, March-April-2021, ISSN (Online): 2395-566X

Fatigue Life Analysis of Tube Flange Welded Joint using ANSYS

M. Tech. Scholar Sumit Kumar, Prof. G.R. Kesheorey (Supervisor & HOD), Dr. A.J. Siddiqui (Executive Director)
Department of ME,
Vindhya Institute of Technology & Science, Indore
Sumit2rishi@gmail.com, grkvitsin@gmail.com

***Abstract*- The fatigue cracks are the one of the major cause of failures in welded joints and it us therefore essential to investigate the dimensional parameters affecting the fatigue characteristics of welded joints. The current research investigates the fatigue life characteristics of tube flange welded joint using techniques of Finite Element Method. The effect of dimensi ons i.e.,h, α and t on fatigue life and safety factor is investigated using Taguchi response surface method. The 3D response surface plots are generated each variable and range of dimensions are evaluated for which safety factor is maximum or minimum. The CAD modeling finite element analysis is conducted using ANSYS software.**

***Keywords*- Weld Joint, Fatigue Life, Safety Factor, ANSYS.**

I. INTRODUCTION

The fatigue cracks are mostly found at weld root after numerous cycles and are difficult to detect. These cracks propagate through the throat thickness. The fatigue failure of weld joints is caused due to various factors which include material, method as well as misalignment of parts for weld and each has varying effect.

It is almost impossible to avoid material imperfections as it is always deformed during welding process. The uneven welds due to poor workmanship results in stress concentrations. Depending on geometry of the joint, the misalignment can be axial or angular in different directions. The stress concentrations created through irregularities on the surface, holes, notches also affect fatigue life of weld.

II. LITERATURE REVIEW

Kumose et al. [1] conducted experimental studies on welded component to determine distortion and effect of pre-straining. The findings have shown that higher flange thickness has higher free angular distortion and the amount of distortion is dependent upon skin stress.

Michaleris et al. [2] conducted experimental testing to determine the effect of thermal tensioning on distortion and residual stress generation on weld joint. The findings have shown that stress could be minimized using thermal tensioning which generates tensile strain at weld toe prior to welding and during welding.

Okerblom et al. [3] proposed model to determine curvature magnitude depending upon heat input from welding using plasticity theory. The model was based on T joint fillet welds and also found that welding cooling phase resulted in permanent deformations. His analytical method is known as mismatched thermal strain method (MTS).

Teng et al.[4] conducted FEA thermal analysis on butt weld to determine the effect of weld conditions and predicted transverse residual stresses. The FEA analysis was conducted for both 10mm spacing and 15mm spacing of stress points.

Teng et al. [5] conducted investigation on T joint fillet welds to predict residual stresses using FEA method and spacing was kept at roughly 3mm for longitudinal and transverse stress. He also investigated the restraint effect on residual stresses generated at weld toe which decreased with restraint. Figure 2 shows the transverse stress perpendicular to the weld line.

III. OBJECTIVE

In the current research we are investigating tube-flange welded joints subjected to torsional loading using ANSYS FEA. The design would be optimized using response surface optimization technique using optimal space filling design. The optimization parameters are geometric which would be h, α and t as shown in figure below. The responses of equivalent stress, shear stress and fatigue life would be generated from this optimization technique along with sensitivities of each optimization variable i.e. h, α and t.

© 2021 IJSRET

International Journal of Scientific Research & Engineering Trends
Volume 7, Issue 2, March-April-2021, ISSN (Online): 2395-566X

IV. METHODOLOGY

The CAD model of geometry is developed as per schematic shown in figure below. The model is developed in ANSYS design modeler using sketch and revolve tool. The dimensions are taken as follows:

Table 1. Dimensions of geometry. [6]

L	1 cm
H	2cm
h	.5cm
t	1cm
α	135^0

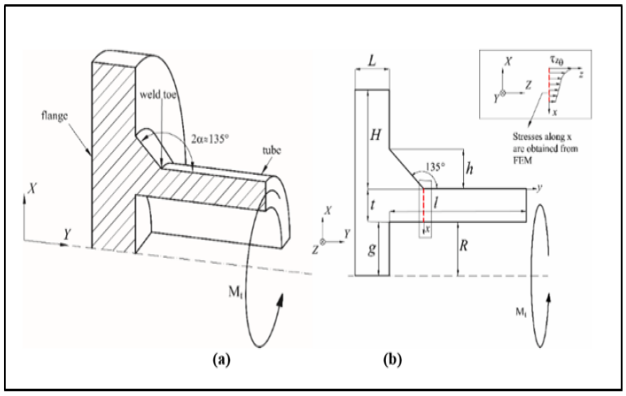

Fig 1. Schematic of tube flange joint.

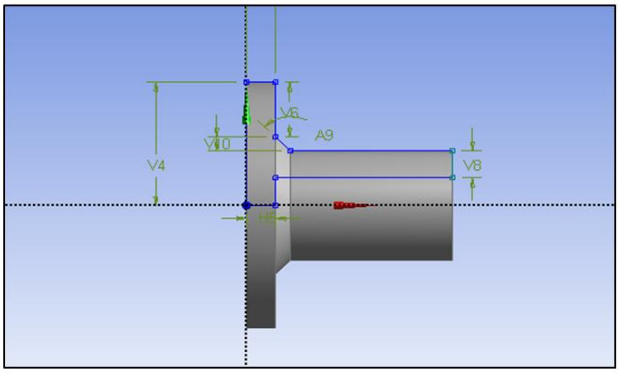

Fig 2. CAD model of tube flange weld joint.

The CAD model of geometry is developed as per literature [6] using ANSYS design modeler. Initially sketch is developed as shown by blue colored cross section in figure 2 above. The dimensions are defined as per table 1 above. The sketch is then revolved to 3600angle to developed full model. The CAD model has sharp angles and edges which makes it complex geometry and therefore it is meshed using tetrahedral elements. The relevance is set to fine, smoothing set to fine, transition ratio .272 and growth rate set to default. The meshed model is shown in figure 3 below.

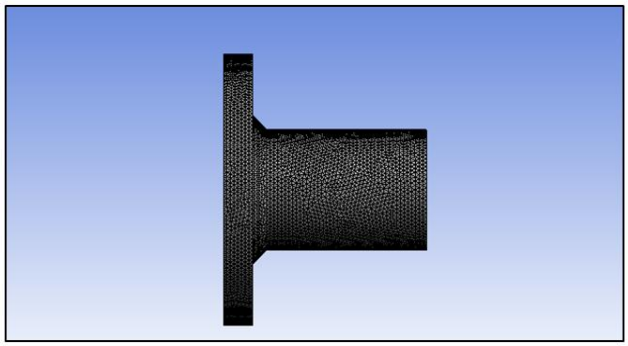

Fig 3. Meshed Model

The CAD model is applied with fixed support at left face of geometry as shown in figure 4 below and rotational moment of 10 N-m on right face as shown in figure 5 below.

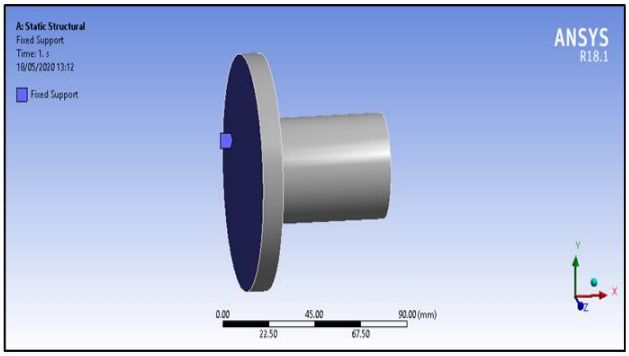

Fig 4. Fixed Support.

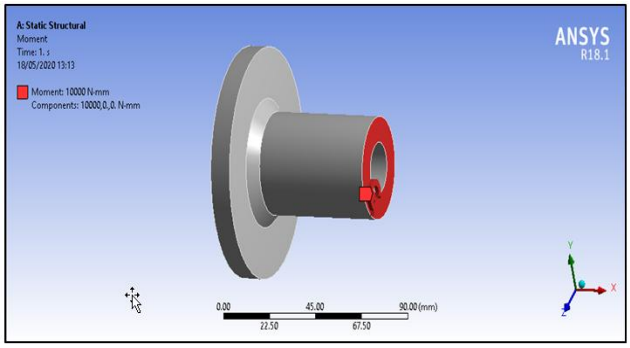

Fig 5. Moment on right face.

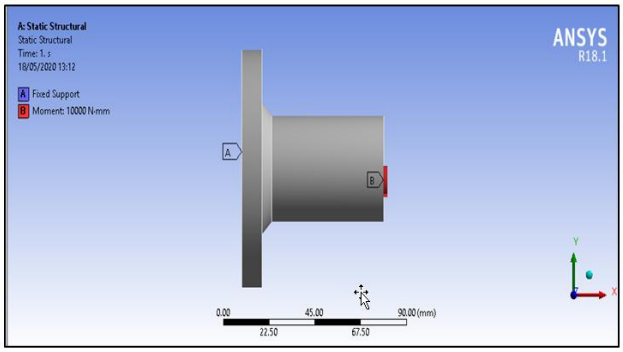

Fig 6. Combined loads and boundary condition.

© 2021 IJSRET

In solution stage, the first step involves element stiffness matrix formulation which are assembled to global matrix. The next step involves matrix inversions, multiplications to get results at nodes which are interpolated for entire element edge length.

V. RESULTS AND DISCUSSION

The fatigue analysis is conducted using stress life approach. The applied load for fatigue life determination is 50N-m. The purpose of conducting fatigue life analysis is to determine critical regions and responses (effect) of optimization parameters on fatigue life of weld joint. The fatigue analysis is conducted using constant amplitude load with fully reversed cycle as shown in figure 7 below. The mean stress correction selected for analysis is Soderbergh.

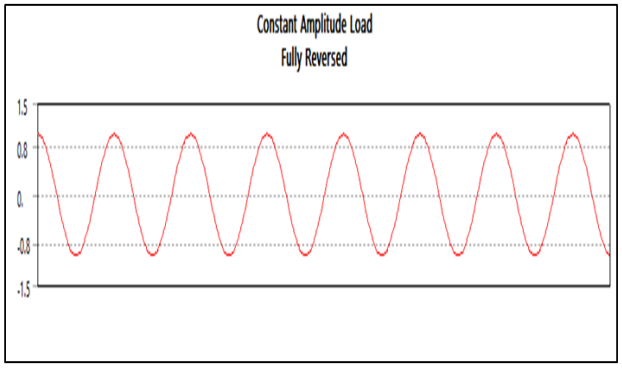

Fig 7. Constant amplitude fully reversed load.

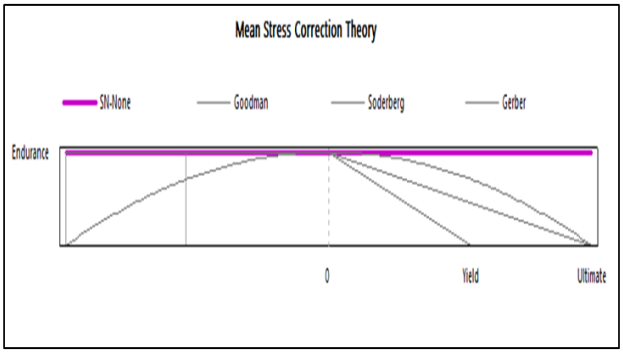

Fig 8. Mean stress correction.

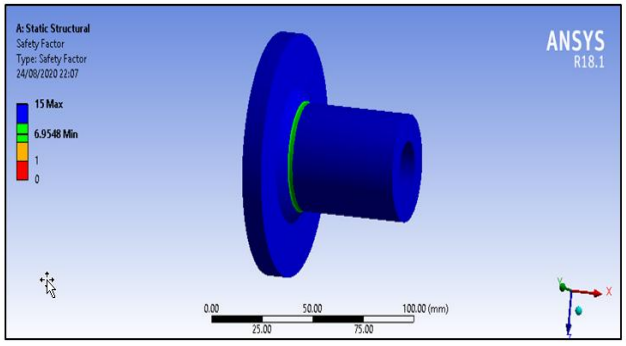

Fig 9. Safety factor plot.

The safety factor plot obtained from FEA analysis shows that highest prone area is joint of flat and cylindrical geometry as shown by green coloured region whereas the other regions shown by dark blue coloured region has highest safety factor.

In other words, we can say that the corner region (shown by green colour) has highest risk of fracture and lowest safety factor of 7.24 under applied loading conditions. The geometry is further optimized using response surface optimization technique. The design points are generated from Taguchi design of experiments are shown in table 2 below.

Table 2. Design points generated from Taguchi Response Surface optimization.

	A	B	C	D	E
1	Name	P5 - Alpha (degree)	P6 - h (cm)	P7 - t (cm)	P9 - Safety Factor Minimum
2	1	135	0.475	1.09	7.3471
3	2	130	0.475	1.09	7.2686
4	3	140	0.475	1.09	7.6254
5	4	135	0.45	1.09	7.5881
6	5	135	0.5	1.09	7.145
7	6	135	0.475	0.98	7.2095
8	7	135	0.475	1.2	7.4428
9	8	130.93	0.45467	1.0006	7.3761
10	9	139.07	0.45467	1.0006	7.6194
11	10	130.93	0.49533	1.0006	6.8555
12	11	139.07	0.49533	1.0006	7.346
13	12	130.93	0.45467	1.1794	7.5297
14	13	139.07	0.45467	1.1794	7.8817
15	14	130.93	0.49533	1.1794	7.0834
16	15	139.07	0.49533	1.1794	7.4867

Using Taguchi response surface (DOE) method involving central composite design scheme various design points are generated. The stress, deformation corresponding to these design points are evaluated from FEA analysis. The compiled results are shown in table 2 above. The maximum and minimum values of safety factor obtained from design of experiments is shown in table 3 below.

Table 3. Maximum and minimum safety factor from DOE.

	A	B	C	D
1	Name	Calculated Minimum	Calculated Maximum	Maximum Predicted Error
2	P9 - Safety Factor Minimum	6.8046	7.9416	0.047084

The responses of h,α and t are evaluated with respect to safety factor and discussed below. The figure 10 below shows safety factor with respect to α which initially decreases up to 131.50 and then increases linearly up to 1400 α value. Therefore, the safety factor variation is direct as well as inverse to α value which highly depends upon the range of α value.

© 2021 IJSRET

International Journal of Scientific Research & Engineering Trends
Volume 7, Issue 2, March-April-2021, ISSN (Online): 2395-566X

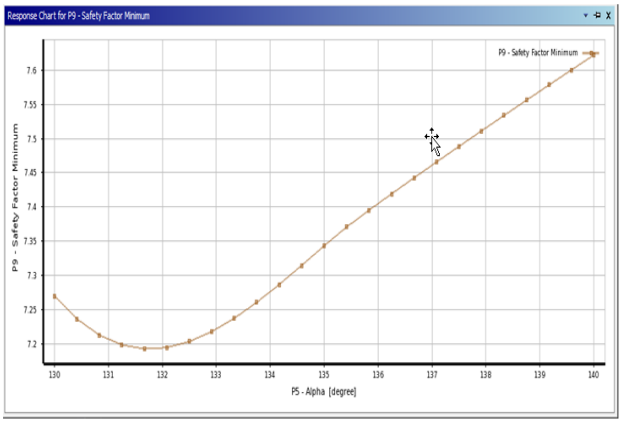

Fig 10. Safety factor vs alpha.

The figure 11 below shows safety factor with respect to h which initially decreases up to .495cm h value and then increases linearly up to .5cm h value. Therefore, the safety factor variation is direct as well as inverse to h value which highly depends upon the range of h value.

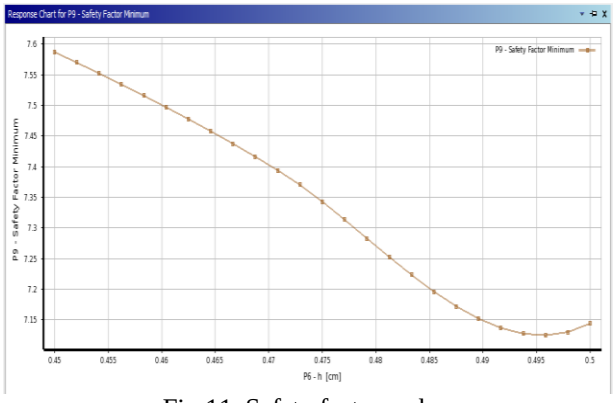

Fig 11. Safety factor vs h.

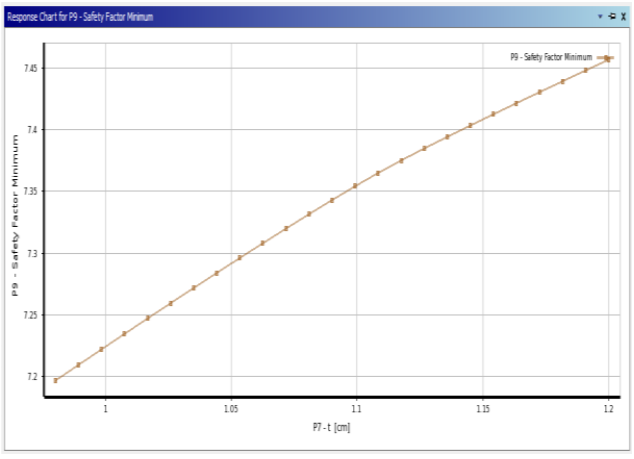

Fig 12. Safety factor vs t.

The figure 12 above shows safety factor with respect to t which initially linearly increases with increase in t value. The minimum safety factor is observed for t value less than 1 and maximum safety factor is observed for t value of 1.2cm.

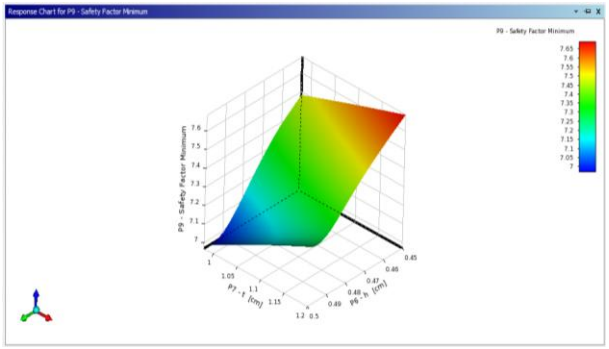

Fig 13. 3D response surface plot of Safety factor vs t and h.

As can be observed from figure 13 above, the maximum safety factor is observed for t values ranging from 1.1cm to 1.2cm and h values ranging from .45cm to .46cm as shown in dark red coloured region. The minimum safety factor is observed for t values ranging from 1.01cm to 1.1cm and h values ranging from .48cm to .5cm as shown in dark blue coloured region.

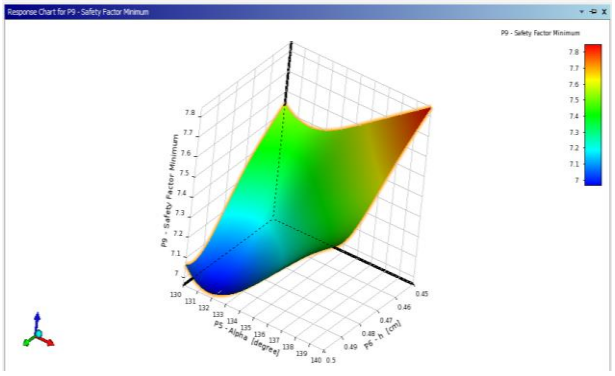

Fig 14. 3D response surface plot of Safety factor vs alpha and h.

As can be observed from figure 13 above, the maximum safety factor is observed for alpha values ranging from 1380 to 1400 and h values ranging from .45cm to 0.46cm as shown by red coloured region. The minimum values is observed for h values ranging from .48cm to .5cm and alpha value ranging from 1300 to 1370 as shown by blue coloured region.

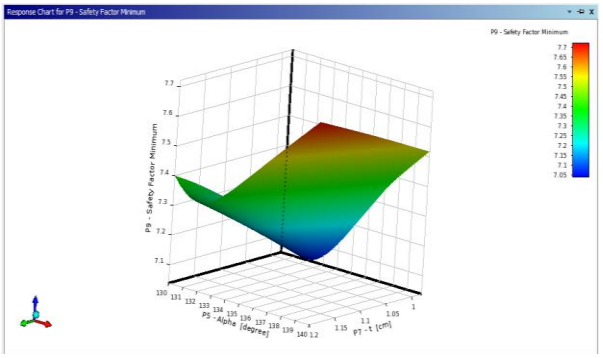

Fig 15. 3D response surface plot of Safety factor vs alpha and t.

© 2021 IJSRET

As can be observed from figure 14 above, the maximum safety factor is observed for alpha values ranging from 1370 to 1400 and t values ranging from 1.1cm to 1.2cm as shown by red coloured region. The minimum values is observed for t values ranging from 1cm to 1.1cm and alpha value ranging from 1300 to 1360 as shown by dark blue coloured region.

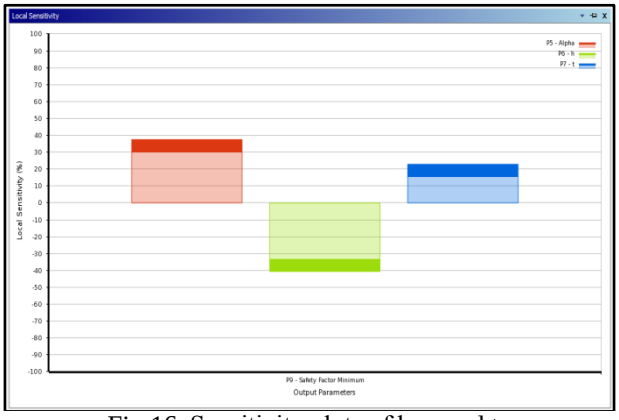

Fig 16. Sensitivity plots of h, α and t.

The sensitivity plot (figure 15) obtained from analysis shows positive sensitivity of alpha and t variable for safety factor which signifies that increasing values of these parameters would increase safety factor and decreasing value of these variables would decrease safety factor.

The sensitivity percentage of alpha is 37.45 (positive) and t is 22.88(positive). The h variable shows negative sensitivity of 40.59(negative) which means increasing this variable value would decrease safety factor and decreasing this variable would increase safety factor. Out of all the 3 variables, the h variable has highest effect on safety factor.

VI. CONCLUSION

The FEA analysis of weld geometry is conducted using ANSYS software and results are analytically verified. The weld parameters of geometry are optimized using design of experiments and response surface methodology. The response surface plots are generated for deformation and shear stress. From response surface plots the range of magnitude of parameters (h, α and t) can be determined for maximum and minimum values of shear stress, deformation and safety factor.

REFERENCES

[1] Kumose, T., T. Yoshida, T. Abbe, and H. Onoue. “Prediction of Angular Distortion Caused by One-Pass Fillet Welding.” Welding Journal. 1954: 945-956.
[2] Michaleris, P., J. Dantiz, and D. Tortorelli. "Minimization of welding residual stress and distortion in large structures." Welding Journal 11 (1999): 361-366s.
[3] Okerblom, N.O. "The Calculations of Deformations of Welded Metal Structures." 1958 (Her Majesty’s Stationery Office, London).
[4] Teng, T., and C. Lin. “Effect of welding conditions on residual stresses due to butt welds.” International Journal of Pressure Vessels and Piping 75 (1998): 857-864.
[5] Teng, T., C. Fung, P. Chang, and W. Yang. Analysis of Residual Stresses and Distortions in TJoint Fillet Welds. International Journal of Pressure Vessels and Piping 78 (2001): 523-538.
[6] Yaoyu Hua (2019), “Notch stress analysis and fatigue strength assessment of tube-flange welded joints under torsion loading” Elsevier, Ocean Engineering 186.

© 2021 IJSRET