

OPTIMIZATION OF WELD JOINT PARAMETERS USING RESPONSE SURFACE METHOD ANSYS

Sanjeev Kumar Sah¹, DR Rajesh Nandal²

Affiliation: PG Student, Department of mechanical Engineering OITM, Hisar Haryana India ¹

Assistant Professor and Head of Department of mechanical engineering, OITM Hisar, Haryana Indian ²

Abstract: The strength and life of weld joint depends on various geometric and environmental factors. The environmental factors influencing weld joint are difficult to control but geometric dimensions affecting strength of weld joint can be optimized. The current research investigates the effect of different design parameters of weld joint on strength and fatigue life of weld joint using Aluminium 6061 material. The design is optimized using response surface optimization technique from CCD (Central Composite Design scheme). The optimization parameters are geometric which is h , a and t . 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 , a and t . The CAD model of weld joint is developed using ANSYS design modeler and FEA analysis is conducted using ANSYS software. Out of the three variables selected for analysis h has highest sensitivity for shear stress and normal stress and therefore should be given highest priority in design of weld joints.

Key Words: FEA, Weld joint, Response Surface Method

1. INTRODUCTION:

The welding process involves joining of two metals at suitable thermophysical conditions and is used widely by many industries. Welding has become the primary means of fabrication or repair of metals and the application it's found in space also. The stresses are induced in weld joints due to repetitive loading. These stresses are highly concentrated at weld toe. The stress concentration may cause damage to structures and therefore requires proper maintenance. Fatigue strength assessment is a critical issue for offshore structures[9]. According to the welding standards EC3 and IIW[10], the nominal stress approach is the most direct method to assess the fatigue strength of steel welded joints. However, it does not consider the local geometrical discontinuities at the weld toe, which cause the assessment results inaccurate.

2. LITERATURE REVIEW

Acevedo, A. Nussbaumer et al. [1] investigated weld joints on bridge tubular joints by residual stress assessment. The residual stresses from findings were completely different

from theoretical values. The non-destructive testing of joint using neutron diffraction technique are also evaluated.

Chattopadhyay et al[2] conducted FEA analysis on weld toe to determine its fatigue characteristics. The author proposed approach for determination of peak stress using the classical stress concentration factors which could be easily implemented for any loading combination.

Cotrell et al.[3] presented the design, analysis, and testing of a two-bladed hub. This section discusses report conclusions and recommendations on design methodology, and testing techniques, as well as work on the hub that remains to be completed. In thesis, author focus on the analysis and testing of the hub body. The experimental and FEA analysis on hub body indicated of its adequate strength to fatigue failures.

Donders S. et al[4] studied the effect of spot weld failure on dynamic vehicle performance in white (BIW) structure. The effect of spot weld failure on vehicle structural

performance, robustness was investigated using techniques of Finite Element Analysis.

Ertas A H. et al [5] conducted FEA fatigue life analysis on spot welds using strain life approach. The findings have shown that number of spot welds, diameter of spot welds, weld to weld distance, weld to edge distance are important parameters affecting weld strength characteristics.

Mustafa Aygul et al.[6] conducted fatigue life analysis of orthotropic bridge deck and studies three types of weld fatigue failures. The system is expected to find wide application in the fields of fatigue design and maintenance for ships, bridges, plants, construction machinery, and other welded structures

Nacy S.M.et al[7] conducted vibrational analysis(modal) on stiffened conical shell with its effect on spot weld. The damping factor, mass and effect of stiffness is also investigated with respect to mode shapes and corresponding natural frequencies.

Yaoyu Hua et al [8] investigated tube flange joint using singularity length method (SLM) and Finite Element Method. A rapid calculation method of the mode III notch stress intensity factor (NSIF K3) without any FEA process is presented based on the SLM. Good agreement is showed between the fine meshed FE, the coarse meshed FE for the strain energy density (SED) and the proposed method.

3. OBJECTIVE

The current research investigates the Al-6061 tube flanges using ANSYS FEA software. The effect of weld joint parameters i.e. h , α and t are analyzed using Taguchi response surface method. The shear stress and deformation are evaluated along with sensitivity. The base design of tube flange weld joint us taken from literature [8].

4. 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 shown in table 1 below and is taken from literature [8]

Table 1: Dimensions of geometry [8]

L	1 cm
H	2cm
h	.5cm

t	1cm
α	135^0

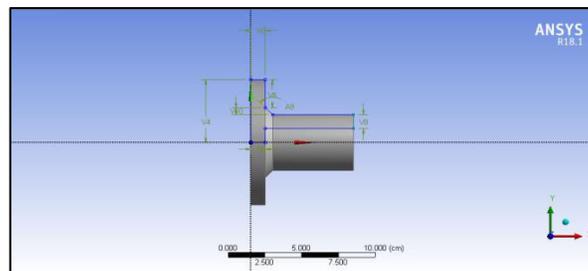


Figure 1: CAD model and sketch of cross section

The CAD model of geometry is developed using ANSYS design modeler. Initially sketch is developed as shown by blue colored cross section in figure 1 above. The dimensions are defined as per table 1 above. The sketch is then revolved to 360^0 angle 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 [11]. The relevance is set to fine, smoothing set to fine, transition ratio .275with growth rate set to default. The tetrahedral element has 4 nodes with 3 degrees of freedom at each node while meshed model is shown in figure 2below.

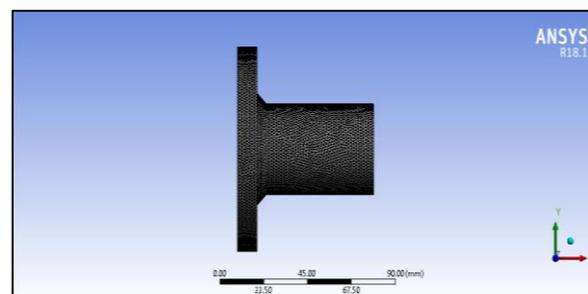


Figure 2: Meshed model

The CAD model is applied with appropriate loads and boundary conditions as shown in figure 3 below. The left face of model is applied with fixed support and right

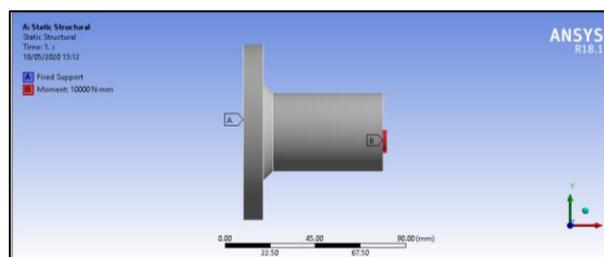


Figure 3: Combined loads and boundary condition

The analysis is run using sparse matrix solver. The results are calculated at nodes and interpolated for entire element edge length.

5. RESULTS AND DISCUSSION

The FEA analysis of Al-6061 tube flanges is conducted and shear stress along with deformation plot is obtained.

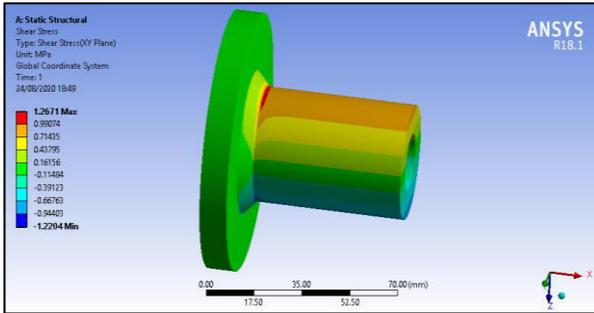


Figure 4: Shear stress distribution plot

The maximum shear stress is developed at corner point as shown in figure 4 above thereby making it highly susceptible to fatigue failure. The magnitude of stress generated at corner is 1.267MPa under given loading conditions.

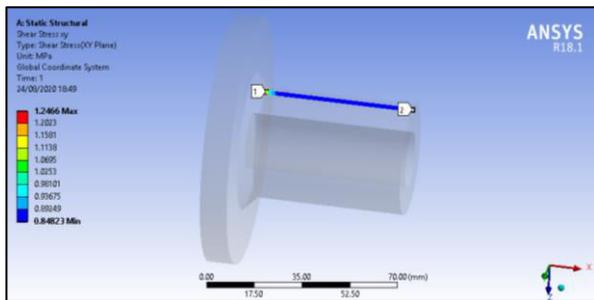
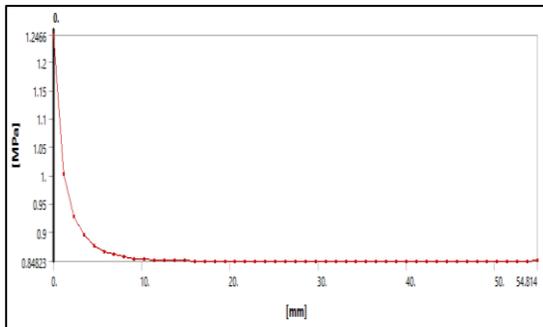


Figure 5: Shear stress along curve (nominal stress distribution)



6: Shear stress vs distance from corner

The shear stress vs distance curve plot given in figure 6 above shown that shear stress decreases on moving away

from corner and becomes minimal towards the end whereas the deformation is highest on open end.

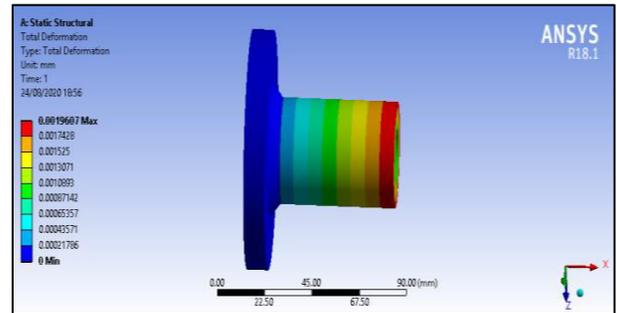


Figure 7: Deformation plot

The deformation contour shows the maximum deformation value at right end of tube flange and minimum at fixed portion of flange as shown by dark blue color. The shear stress vs alpha plot shown in figure 8 below shows fluctuating behavior of stress w.r.t alpha. The shear stress decreases up to 132.5° alpha value and then increases up to 138.5° alpha value.

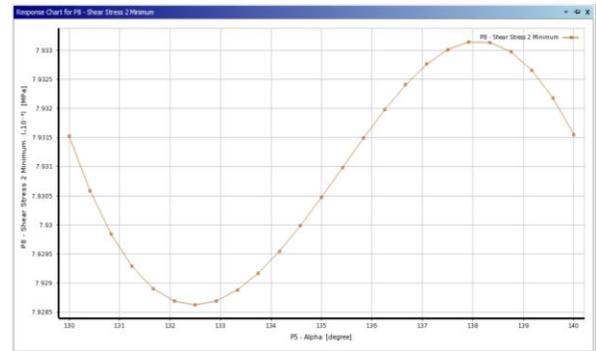
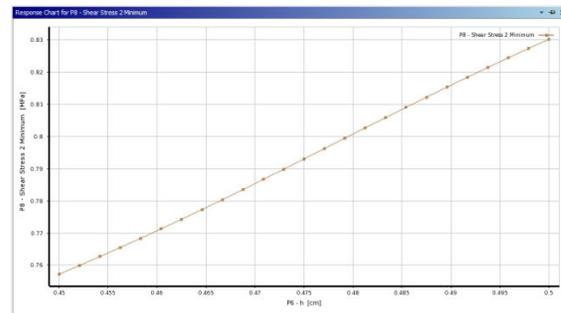


Figure 8: Shear stress vs alpha



9: Shear stress vs h

The shear stress value increases linearly with increase in h as shown in figure 9 above. The minimum shear stress is observed for h value of .45cm and maximum shear stress is observed for h value of .5cm.

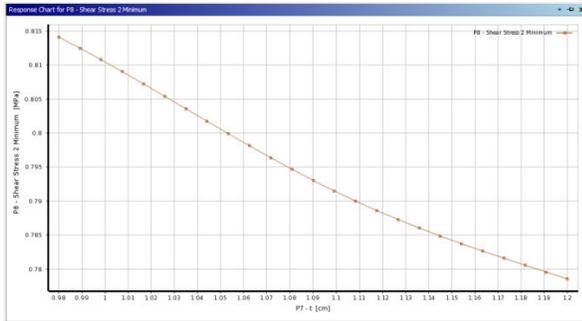


Figure 10: Shear stress vs t

The shear stress value decreases linearly with increase in t value as shown in figure 10 above. The minimum shear stress is observed for t value of 1.2cm and maximum shear stress is observed for t value of .98cm.

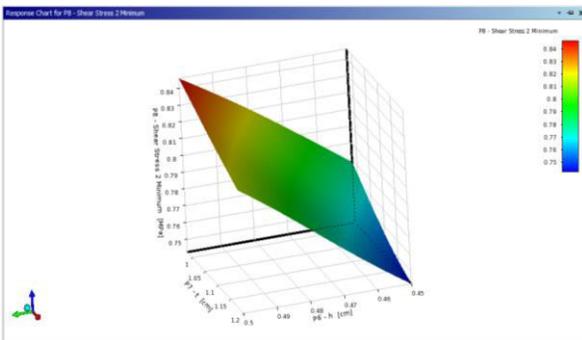


Figure 11: 3D response surface plot of shear stress vs h and t

As can be observed from figure 11 above, the maximum shear stress is observed for t values lower than 1.05cm and h values ranging from .49cm to .5cm as shown in dark red colored region. The minimum values is observed for t values ranging from 1.1cm to 1.2cm and h values ranging from .47cm to .45cm as shown in dark blue colored region.

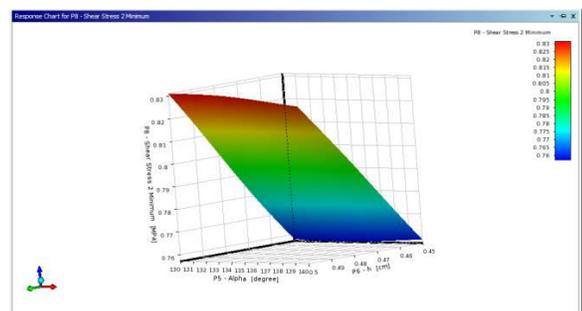


Figure 12: 3D response surface plot of shear stress vs alpha and h

As can be observed from figure 12 above, the maximum shear stress is observed for alpha values ranging from 130° to 140° and h values ranging from .49cm to 0.5cm as shown

by red colored region. The minimum values is observed for h values ranging from .45cm to .47cm and alpha value ranging from 130° to 140° as shown by blue colored region.

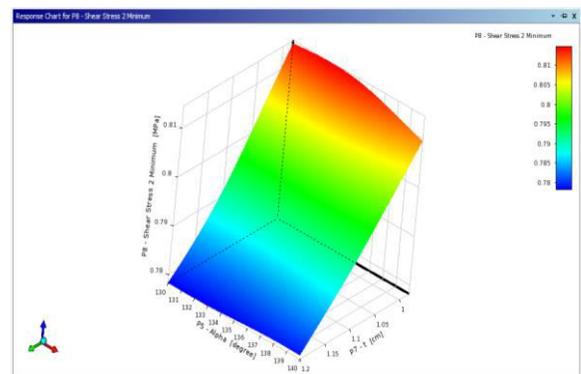


Figure 13: 3D response surface plot of shear stress vs alpha and t

As can be observed from figure 13 above, the maximum shear stress is observed for alpha values ranging from 130° to 138° and t values less than 1cm as shown by red colored region. The minimum values is observed for t values ranging from 1.15cm to 1.2cm and alpha value ranging from 130° to 140° as shown by dark blue colored region.

5. CONCLUSION

The FEA analysis of weld joint is conducted using ANSYS software to determine shear stress and deformation. The Taguchi response surface optimization is conducted and responses of various optimization parameters are evaluated. The sensitivity plot 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.

REFERENCES

[1]C.Acevedo, A. Nussbaumer, Residual stress estimation of welded tubular k-joints under fatigue loads, Swiss Federal Institute of Technology, Lausanne, Switzerland, pp 3-6(2009).
 [2]A.Chattopadhyay, G. Glinka, M. El-zein, J. Qian and R.Formas, Stress analysis and fatigue of welded structures,

Welding in the world Peer Reviewed Section, Vol 55,pp.9-15 (2013).

[3] J. Cotrell,(2002), The mechanical design, analysis, and testing of a two-bladed wind turbine hub, Technical Report, NREL/TP-500-26645, pp.56-66.

[4] Stijn Donders, Marc Brughmans, Luc Hermans, Nick Tzannetakis “The Effect of spot Weld Failure on Dynamic Vehicle Performance”. LMS International, Interleuvenlaan 68 (2005), 3001 Leuven, Belgium.

[5] Ahmet H.Ertas a, Fazıl O.Sonmez “Design optimization of spot-welded plates for maximum fatigue life” Finite Elements in Analysis and Design 47 (2011) 413–423.

[6] Mustafa Aygul,(2012), Fatigue analysis of welded structures using the finite element method, Department of Civil and Environmental Engineering Division of Structural Engineering, Steel and Timber Structures Chalmers University of Technology, Gothenburg, Sweden, ISSN no. 1652-9146,pp.35-60

[7] S. M. Nancy ,N. K. Alsahib, F. F. Mustafa “ Vibration Analysis of Plates With Spot Welded Stiffeners” JJMIE Volume 3, Number 4, December 2009.

[8] Yaoyu Hua, “Notch stress analysis and fatigue strength assessment of tube-flange welded joints under torsion loading” Ocean Engineering 186(2019) Elsevier

[9] Ma, Y., Han, C., Qu, X., 2018. Fatigue assessment method of marine structures subjected to two Gaussian random loads. Ocean. Eng. 165, 107–122.

[10] Hobbacher, A., 2016. Recommendations for Fatigue Design of Welded Joints and Components. IIW Collection. Springer International Publishing

[11] ANSYS Solver Modelling Guide 2003-01-25