# INVESTIGATIONS OF FATIGUE LIFE ON WELD JOINT PARAMETERS USING FEA AND TAGUCHI DESIGN OF EXPERIMENTS

**Rahul Dev[1], Gora Ram Sharma[2*]**
[1, 2*]Bhai Maha Singh College of Engineering, Sri Muktsar Sahib

**Abstract:** Full-penetration steel tube-flange welded joints (TFWJs) are widely used to connect tubes and plates. The geometrical discontinuities at weld toe corners always lead to localized stress concentrations, which drive fatigue cracks initiating and propagating with alternating loads. 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 Optimal Space Filling design scheme. The optimization parameters are geometric which is h, α and t. The responses of fatigue life would be generated from this optimization technique along with sensitivities of each optimization variable i.e. h, α and t. The CAD model of weld joint is developed using ANSYS design modeller and FEA analysis is conducted using ANSYS software. The 3D response surface plots are generated for different variables to determine range of dimensions for which stresses and fatigue life are minimum or maximum. The sensitivity plots are also generated which depicts the sensitivity of each variable and which variable has highest effect on output parameters. 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.
**Keywords:** FEA, Weld joint, Response Surface Method, Taguchi Design of Experiments

## I. INTRODUCTION

The welded joints usually suffer from various welding deformation patterns such as angular distortion, longitudinal and transverse shrinkage in fabrication of structural members in ship building, automobile and other industries. The angular distortion is clarified through numerical calculation in all these applications. The stresses in the weldment are evaluated by varying the gap between the parent plates which may occur during manufacturing. It is a type of mechanical joints like rivets and bolt used in ancient times. The residual stress in a welded joint is compared by carrying out computations by making 3D models and 2D models.

A mathematical model can easily predict the weld penetration as a function of welding process parameters. The constrained optimization method is then applied to this model to optimize process parameters for maximizing weld penetration.

Welding has many merits over bolting and riveting. Welding permits direct stress between members eliminating gusset and splice plates necessary for bolted structures. So that joint weight is decrease. The elimination of holes increases efficiency when we consider tension member as factor. For this fabrication cost is less because of lesser parts, and operations, it reduces labour and economy. Usually welded joints are stronger than the base metal. While considering welded connection stress concentration effect is less. Skilled manpower and inspection is required for welding. Due to environmental conditions or location welding may be difficult. Under fatigue load welded joint obtain cracks due to improper welding.

## II. LITERATURE REVIEW

Ertas et al. [1] 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, and weld to edge distance are important parameters affecting weld strength characteristics. Moshayedi et al. [2] conducted

2D FEA thermal analysis on weld joint to analyze nugget formation at various temperatures. The analysis conducted was coupled thermal, mechanical and parameters of interest were welding temperature. Nacy et al. [3] conducted vibration analysis (modal) on stiffened conical shell with its effect on spot weld. The damping factor, mass and effect of stiffness are also investigated with respect to mode shapes and corresponding natural frequencies. Prasad et al. [4] conducted FEA vibration analysis on spot weld of stiffener plates using Hypermesh software. The findings of FEA results i.e. frequency was validated with experimental results using FFT analyser. The findings have shown that natural frequencies are affected by stiffener profile and pattern and therefore these 2 variables need considerations. Rusinski et al. [5] conducted FEA analysis on spot welds under compressive loads considering geometric and material nonlinearities. The findings have shown that pitch of spot weld and non-linearities has significant effect on strength and fatigue life of spot welds which is either directly proportional or vice versa. Palmonella et al. [6] conducted studies on CWELD and ACM-2 spot weld structures with respect to natural frequencies. The paper also proposed various guidelines for applications in automotive bodies (BIW) model. The structural dynamics of entire body is heavily affected by these spot welds. Hua et al. [7] proposed SLM (singularity length method) for estimation of shear stress distribution of tube flange welded joint. The results of SLM are validated with FEA results. The SIF's (Stress Intensity Factor) is also evaluated from fatigue life analysis. The findings have shown that shear stress distribution is affected by thickness of the flange L, the weld size h, and thickness of the tube t.

## III. TAGUCHI METHOD

Taguchi method is a systematic application of design of experiment technique for the purpose of designing and improving product quality. Optimization of process parameters is the key step in the Taguchi method to achieve high quality without increasing cost. This is because optimization of process parameters can improve quality characteristics and the optimal process parameters obtained from the Taguchi method are insensitive to the variation of environmental conditions and other noise factors. Classical process parameter design is complex and not an easy task. Large number of experiments have to be carried out when the number of process parameters increases. To solve this task, the Taguchi method uses a special design of orthogonal arrays to study the entire process parameter space with a small number of experiments only. After conducting experiments, the optimal test parameter configuration within the experiment design must be determined. To analyze the results, the Taguchi method uses a statistical measure of performance called signal-to-noise (S/N) ratio borrowed from electrical control theory. The S/N ratio developed by Taguchi is a performance measure to choose control levels that best cope up with noise. The S/N ratio takes both the mean and the variability into account. In its simplest form, the S/N ratio is the ratio of the mean (signal) to the standard deviation (noise). The S/N equation depends on the criterion for the quality characteristic to be optimized. The value of the loss function is further transformed into signal-to-noise ratio (S/N). There are three categories of the quality characteristic in the analysis of the S/N ratio, i.e. lower-the-better, higher-the-better and normal-the-better. The S/N ratio for each level of process parameters is computed based on the S/N analysis. Regardless of the quality of the quality characteristic, a large S/N ratio corresponds to a better quality characteristic. Therefore, the optimal level of the process parameters is the level with the highest S/N ratio.

## IV. FEA SIMULATION

The easiest way to determine the process parameters for the welding process is to model the welding processes using FEA software's and simulate it. The FEA software such as WELDSOFT, ABACUS etc. are more user-friendly to the designer such that the modification of any data is easily done on seeing the preview of the simulated weld conditions. The accuracy of the result depends upon the assumptions made in constructing the model. The results obtained from FEA simulation are verified by running a confirmatory experiment.

FEA method is used to model the welded joints, to determine the time-temperature relationship, analyze displacement of welded model under load and to determine the stresses acting throughout the structure. The results are useful to estimate the safe conditions of the welded structure. Wen et al. [8] modeled SAW process to investigate the heat transfer characteristics in the fusion and heat affected zone during welding using Finite element package ABACUS. Modeling and finite element analysis on heat transfer of GTAW arc weld pool using finite element analysis software was done by Fenggui [9]. Thermal and thermo mechanical analysis of friction stir welding using Finite element modeling was developed by Chen et al. [10]. FEM is a numerical method used to solve various problems of different disciplines such as steady, transient, linear, non-linear analysis, heat transfer, fluid flow, electromagnetism etc. Teng et al. [11] studied the effect of weld geometry and residual stresses on fatigue in butt-welded joints using the finite element analysis procedure. Shahram et al. [12] employed FEM using ABACUS as an efficient approach to compute the residual stresses in welded joints. The finite element program ABAQUS was used by Cho et al. [13] to determine the fatigue strength in laser welding of the lap joint.

## V. METHODOLOGY STEPS

- **CAD Modeling**

The CAD model of geometry is developed as per schematic shown in figure 1. The model is developed in ANSYS design modeler using sketch and revolve tool. The CAD model of geometry is developed as per literature [1] using ANSYS design modeler. Initially sketch is developed as shown by blue colored cross section in figure 2. The sketch is then revolved to $360^0$ angle to developed full model.

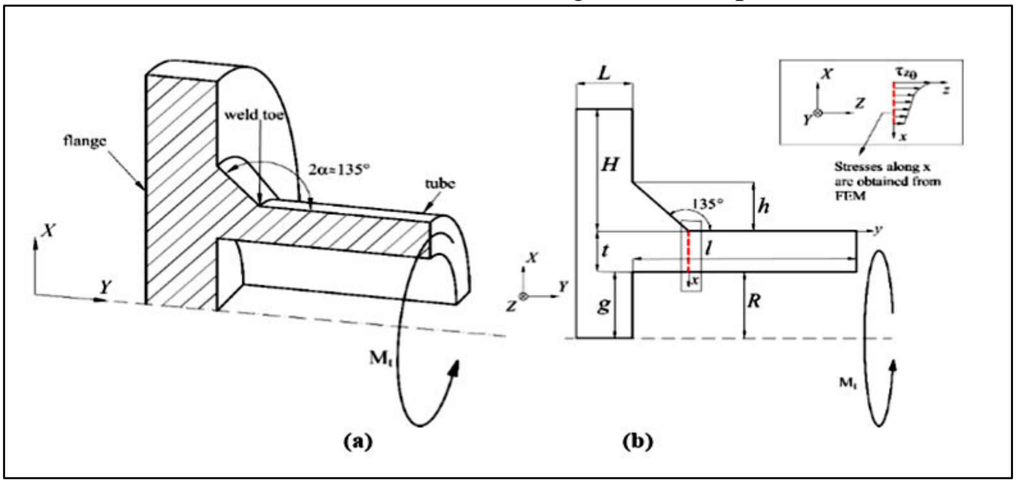



Figure 1: a) Quarter model b) Parameter definition [14]

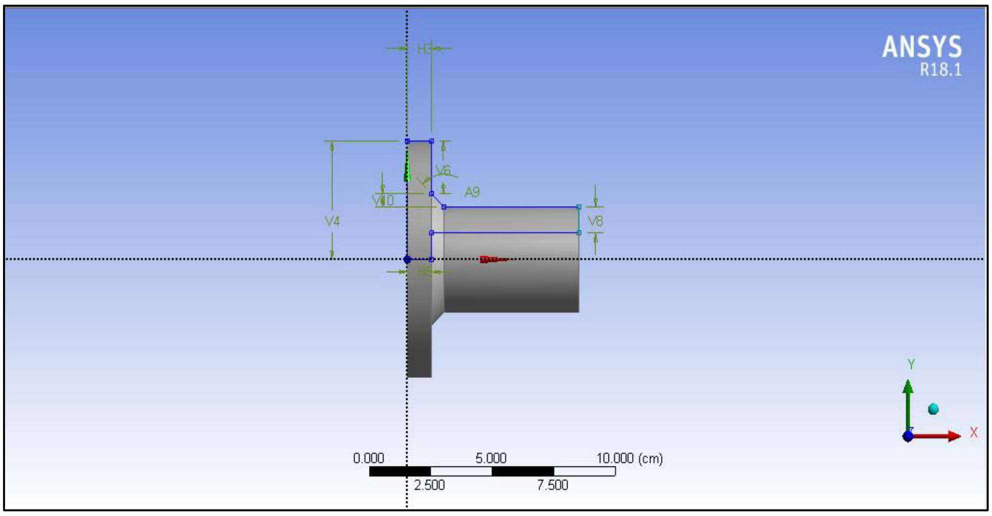



Figure 2: CAD model using ANSYS Design Modeler [14]

- **Meshing**

The CAD model has sharp angles and edges which make it complex geometry and therefore it is meshed using tetrahedral elements. The relevance is set to fine, smoothing set to fine, transition ratio 0.272 and growth rate set to default. The tetrahedral element has 4 nodes with 3 degrees of freedom at each node as shown in figure 4.3

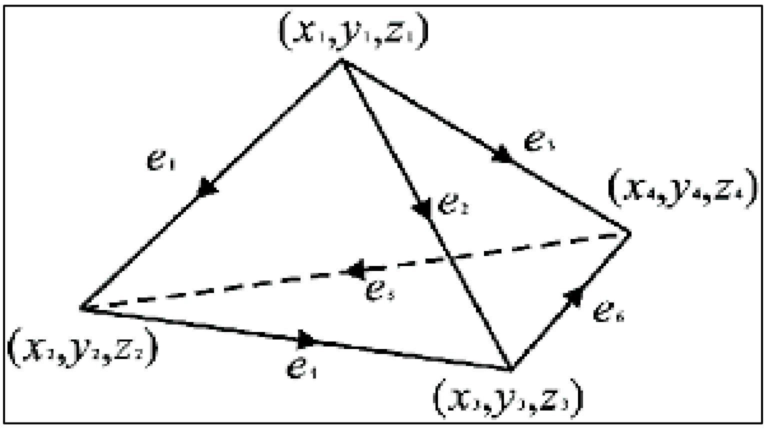



Figure 3: Tetrahedral Element Shape

- **Loads and Boundary Conditions**

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

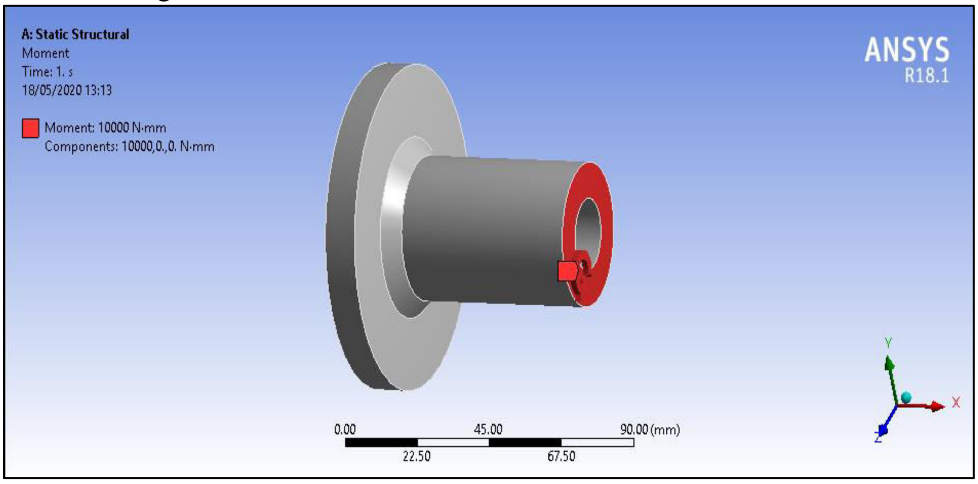



Figure 4: Moment on Right Face

- **Solution stage**

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

- **Response Surface Optimization**

The weld joint is optimized using techniques of RSM and variables selected for optimizations are h, α and t. These optimization variables influence output parameters i.e. normal stress, deformation and shear stress. The goodness of fit curves is developed for the optimization and sensitivity plots are also generated.

## VI. RESULTS & DISCUSSIONS

**FEA analysis for fatigue life determination**

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.

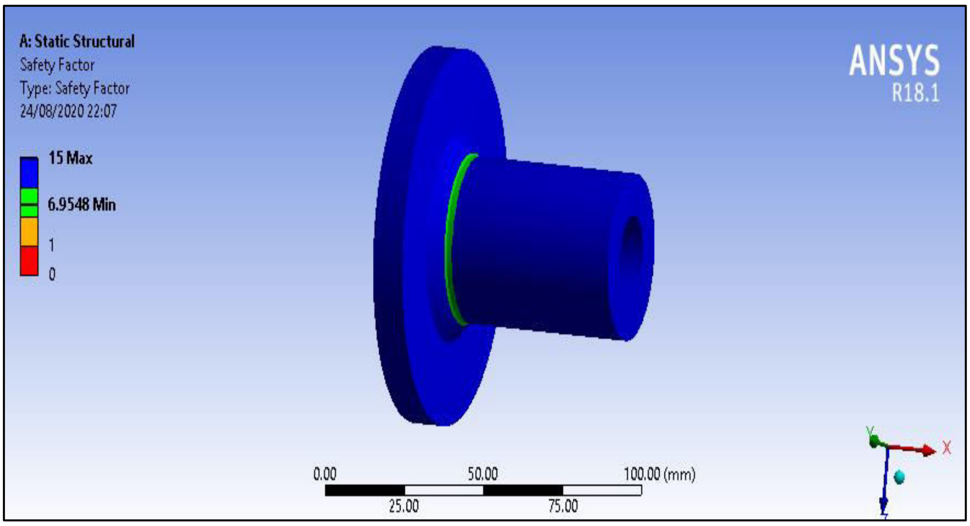



Figure 5: Safety factor plot

The safety factor plot obtained from FEA analysis shows that highest prone area is joint of flat and cylindrical geometry as shown by green colored region whereas the other regions shown by dark blue colored region has highest safety factor. In other words, we can say that the corner region (shown by green color) 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 is shown in table 1.

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 1. The responses of h, α, and t are evaluated with respect to safety factor and discussed below

Table 1: 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 |

The figure 6 shows safety factor with respect to α which initially decreases up to $131.5^0$ and then increases linearly up to $140^0$ 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.

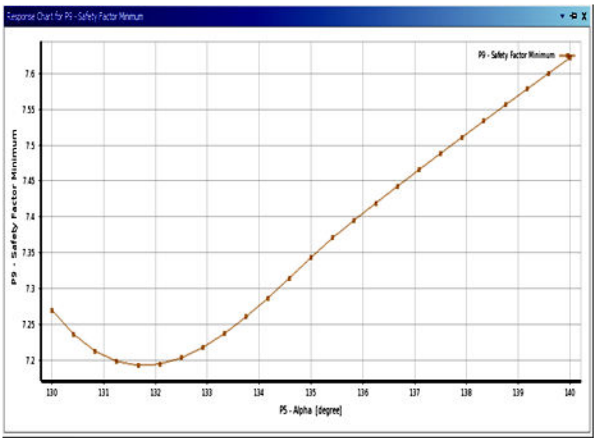



Figure 6: Safety factor versus α plot

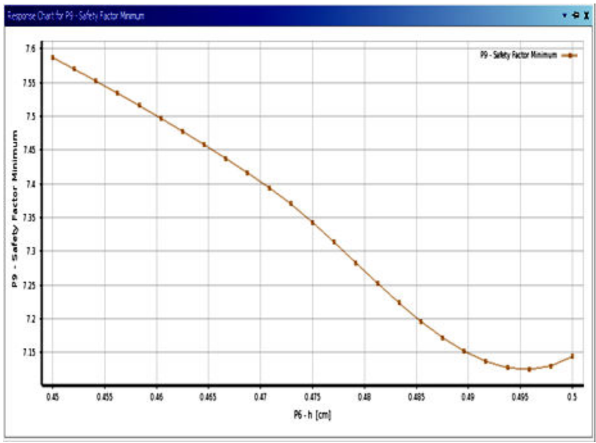



Figure 7: Safety factor versus h plot

The figure 7 shows safety factor with respect to h value. Safety factor initially decreases up to 0.495 cm h value and then increases linearly up to 0.5 cm 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.

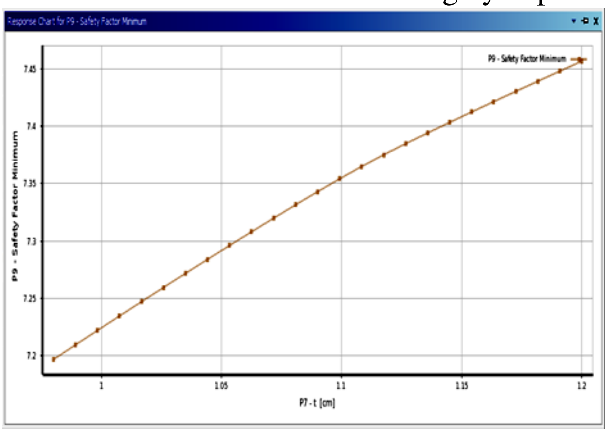



Figure 8: Safety factor versus t plot

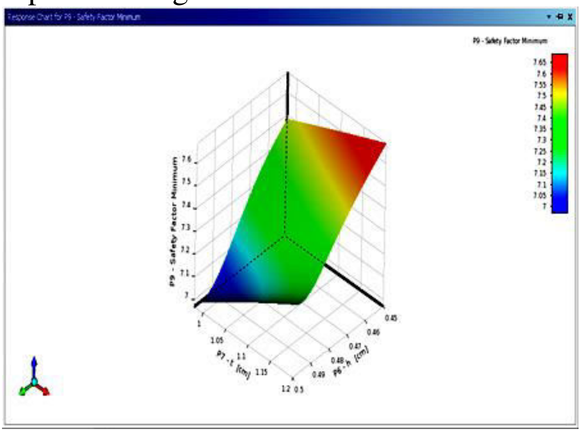
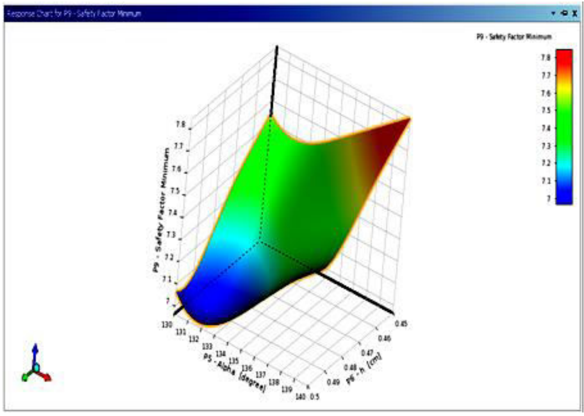



Figure 9: 3D response surface plot of Safety factor versus t and h

The figure 8 shows safety factor with respect to t value. Safety factor initially increases linearly 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 1.2 cm. As can be observed from figure 9, the maximum safety factor is observed for t values ranging from 1.1 cm to 1.2 cm and h values ranging from 0.45 cm to 0.46 cm as shown in dark red colored region. The minimum safety factor is observed for t values ranging from 1.01 cm to 1.1 cm and h values ranging from 0.48cm to 0.5cm as shown in dark blue colored region.



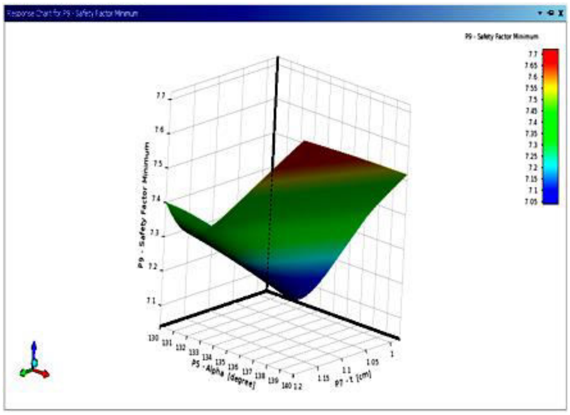


Figure 10: 3D response surface plot of Safety factor versus α and h

Figure 11: 3D response surface plot of Safety factor versus α and t

As can be observed from figure 10, the maximum safety factor is observed for α values ranging from $138^0$ to $140^0$ and h values ranging from 0.45 cm to 0.46 cm as shown by red colored region. The minimum values is observed for h values ranging from 0.48 cm to 0.5 cm and α value ranging from $130^0$ to $137^0$ as shown by blue colored region. As can be observed from figure 11, the maximum safety factor is observed for α values ranging from $137^0$ to $140^0$ and t values ranging from 1.1 cm to 1.2 cm as shown by red colored region. The minimum values is observed for t values ranging from 1 cm to 1.1 cm and α value ranging from $130^0$ to $136^0$ as shown by dark blue colored region. Figure 12 provide a sensitivity plot for the three weld parameters i.e. h, t, and α.

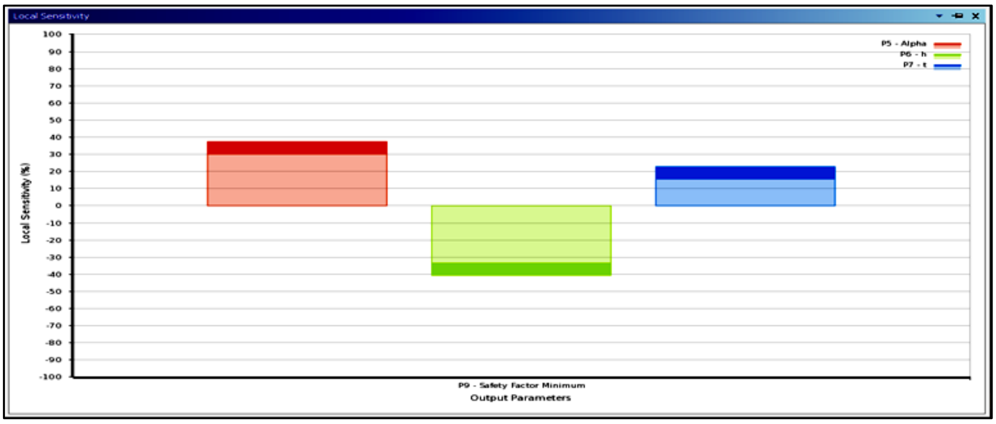



Figure 12: Sensitivity plots of h, α and t

## CONCLUSIONS

The present work as described in this paper is an attempt to understand the concept of Taguchi design of experiments and Finite Element Analysis (FEA) for calculating the fatigue life of weld parameters which include h, t, and α. Firstly, FEA model is designed for analyzing the safety and sensitivity response of fatigue life on different weld parameters. Then various plots are designed to analyze these effects on weld parameters. The sensitivity plot obtained from analysis shows positive sensitivity of α 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 α 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 three variables, the h variable has highest effect on safety factor.

## REFERENCES

[1] A. H. Ertas, A. Fazıl, O. Sonmez, "Design optimization of spot-welded plates for maximum fatigue life," Finite Elements in Analysis and Design, 47, pp. 413–423, 2013.

[2] H. Moshayedi and I. Far, "Numerical and experimental study of nugget size growth in resistance spot welding of austenitic stainless steels," Journal of Materials Processing Technology, 212, pp. 347– 354, 2012.

[3] S. M. Nacy, N. K. Alsahib and F. F. Mustafa, "Vibration Analysis of Plates with Spot Welded Stiffeners," JJMIE, Vol. 3, No. 4, pp. 202-206, 2019.

[4] K. Prasad and P. N. Jadhav, "Vibration Analysis Plates with Spot Welded Stiffeners," International Journal of Latest Trends in Engineering and Technology (IJLTET), Vol. 7, No. 11, pp. 103-109, 2014.

[5] E. Rusi´nskia, A. Kopczy´nskib and J. Czmochowskia, "Tests of thin-walled beams joined by spot welding," Journal of Materials Processing Technology, 157–158, pp. 405–409, 2014.

[6] M. Palmonella and M. Friswell, "Improving spot weld model in structural dynamics," ASME 2003 Design Engineering Technical Conferences and Computers and Information in Engineering Conference Chicago, Illinois, USA, September 2-6, 2003.

[7] Y. Hua, "Notch stress analysis and fatigue strength assessment of tube-flange welded joints under torsion loading," Elsevier, Ocean Engineering, 186, pp. 207-213, 2019.

[8] S.W.Wen, P. Hilton and D.C. I Fariugia, Finite element modeling of Submerged arc welding process, Journal of Material Processing Technology,1 19, 2001, 203-210

[9] Fenggui Lu, Shun Yao, Songnian Lou and Yongbing Li, Modeling and finite element analysis on GTAW arc and weld pool, Computational Materials Science Volume 29, Issue 3 , 2004, 371-378.

[10] C.M. Chen and R. Kovacevic, Finite element modeling of friction stir welding—thermal and thermomechanical analysis, Optics & Laser Technology, Volume 37, Issue 1 , 2005, 33-42.

[11] Tso-Liang Teng, Chin-Ping Fung and Peng-Hsiang Chang, Effect of weld geometry and residual stresses on fatigue in butt-welded joints, International Journal of Pressure Vessels and Piping 79, 2002, 467—482.

[12] Shahram Sarkani, Vesselin Tritchkov, George Michaelov, An efficient approach for computing residual stresses in welded joints, Finite Elements in Analysis and Design 35 ,2000, 247-268.

[13] S.-K.Cho ,Y.-S.Yang ; ,K.-I.Son and I.-Y.Kim, Fatigue strength in laser welding of the lap joint, Finite Elements in Analysis and Design, 40, 2004, 1059 —1070.

[14] C. Olsson, 2005, "Constructions handbook," 3rd edition, Lidingo: Industrial Literature.