PREDICTION OF FATIGUE LIFE OF FULL PENETRATION WELD JOINT USING THERMAL ELASTO PLASTIC PROPERTIES INCORPORATING RESIDUAL STRESS EFFECT

Gokula Krishnan BALASUBRAMANIAN[1], Venkatachalam GOPALAN[2], Atul DESHMUKH[3]

This study presents the finite element analysis of the weld joints used in the off-highway vehicles. A coupled field thermal and static analysis is performed to find the residual stresses in weld and its effect on the fatigue life of the double plate full penetration weld joint. The Design of Experiments approach, via Response Surface Methodology (RSM), is used considering the influence of the parameters like Plate Thickness, Load and Prep angle on the fatigue life. The central composite design, a model in RSM is used to minimize the number of simulations. The Analysis of Variance (ANOVA) is carried out to find the influences of Plate Thickness, Load and Prep angle on the fatigue life. The Regression Equation, obtained through ANOVA, helps to understand the influences of parameters on fatigue life. Later the regression equation is validated with finite element results. The Optimization study is carried out to find optimized parameters for maximum fatigue life. Confirmation simulation using finite element analysis reveals the integrity of optimization study.

Keywords: Residual Stress, Fatigue Life, Moving Heat Source, Welded joints, Design of Experiments, Finite element Method.

1. Introduction

The welded joints are subjected to residual stress due to the heat input applied during the weld. Due to the residual stress, the model is deformed or distorted based on the geometry of the joint and applied loading conditions. This residual stress can be minimized and removed from the welded joint by several methods like cold treatment etc. In off highway vehicles, the welded joints are subjected to repetitive loading conditions. The most common failure in these types of joints is fatigue failure. The residual stress of the weld plays important role in the fatigue life material. It is usually minimized to get better fatigue life when subjected to cycling loading conditions.

[1] Stud., SMEC, Vellore Institute of Technology, Chennai Campus, India, e-mail: iamgokul2110@gmail.com
[2] Prof., SMEC, Vellore Institute of Technology, Chennai Campus, India, e-mail: g.venkatachalam@vit.ac.in
[3] General Manager, Centre of Expertise, JCB India Ltd., Pune, India, e-mail: atul.deshmukh@jcb.com

Amudha et al. [1] performed finite element moving heat source simulation on IS-2062 Grade-B low carbon steel based material and estimated the residual stress formed during the single and double layer deposition of SS-309Mo and Inconel-625. They suggested that the lower residual stress is helpful in preventing cracking and corrosion of the material. Ahmad et al. [2] evaluated the structural reliability of the weld through assessment of fatigue life and residual stress analysis of the weld. The residual stress distribution in transverse and longitudinal direction of the joint was estimated and it was seen that the stress ranges without considering the residual stress effects were 14% higher than the stress ranges with considering the residual stress effects. Samad et al. [3] performed a finite element simulation of welding of aluminum alloy 6082. The authors investigated the effect of welding speed and heat input on weld pool shape and temperature distribution. Ranjbarnodeh et al. [4] proposed a three-dimensional finite element model to estimate the heat transfer during the Gas Tungsten Metal Arc Welding (GTAW) of the stainless steel (SS304) material. They concluded that the distance in which the precipitation is seen in austenite grain boundaries increases, when the heat input is increased. Sagar et al. [5] attempted to simulate the Tungsten Inert Gas (TIG) welding on the titanium G5 alloy (Ti-6Al-4V). It was concluded that the welding speed and temperature has inverse relationship.

Deshmukh et al. [6] predicted the fatigue life of doubler plate weld joint which is used in off-highway vehicle structures. The authors concluded that the prep angle of the weld shows considerable effects i.e., up to 45 degree, the fatigue life of the welded joint increases while fatigue life is reduced after 48 and 60 degrees of prep angles. Deshmukh et al. [7] performed finite element simulation to study the effect of weld penetration on the fatigue life. They observed that there is a change in fatigue life of the material when the weld penetration levels are changed. They also concluded that a lack of penetration would cause a substantial stress increment. Teng et al. [8] made an effective procedure to combine residual stress of weld joints with strain-based fatigue life methods along with thermo elastic plastic theory and finite element method. They concluded that, on increasing the radius of weld toe and decreasing the flank angle, the fatigue life of butt weld could be increased. Li et al. [9] derived a novel approach of moving heat source model for the simulation of residual stress in T joint welding. The authors concluded that the maximum residual stress is seen at the weld toe region and there is no residual stress seen at the center of the weld. The residual stresses were compressive on the both sides of the weld and it was tensile inside the weld. Li et al. [10] investigated the effects of loading conditions, weld geometry and the boundary conditions on the fatigue life of a ferrite-pearlite steel lap joint using finite element method. They concluded that the reduction in the size of finite element model has a significant effect on final results.

Islam and Hassan [11] performed experiments and finite element simulation to investigate the manufacturing and welding residual stress formed in a elbow geometry under low cycle fatigue conditions. The authors found that the manufacturing and residual stresses in the elbow geometry have very less impact on the fatigue life under low cycle fatigue conditions. Peric et al. [12] investigated the effect of the interpass time and various preheat temperatures on the residual stress fields and structural deflections in T-joint weld. It was concluded that the increase in interpass time increases the deflection in the plates. Ramos et al. [13] investigated the configurations of T-joint weld and considered arc welding and laser welding processes. They concluded that the laser beam welding has a reduced heat affected zone as compared with the arc welding process where multiple passes are performed which increases tensile residual stress. Wang et al. [14] investigated the effect of residual stress on fatigue life of T joint weld. They found that the presence of residual stress values leads to decrement in the ductility and it changes the direction of the initiation of the fatigue crack. Cui et al. [15] used an elastic fracture mechanics approach. The authors considered weld residual stress (WRS) and weld residual stress relaxation (WRSR) for simulation. The models had maximum error of 14 % on fatigue considering WRS and WRSR effects. The fatigue life was 17% underestimated when the WRS is considered and WRSR is neglected. At the same time the fatigue life was over estimated by 49% when the WRS and WRSR were together neglected. Deshmukh et al. [18] carried out finite element and experimental investigations on joggle type weld joint using virtual strain gauge. The results were validated along with the Design of experiments approach and was concluded that the applied load and plate thickness show substantial effect on normal stress and fatigue life as compared to the root gap.

From the literature, it is inferred that the influence of residual stress on fatigue life prediction of weld joints used in off-highway vehicle structure is not investigated for weld joints which involves welds in complicated structures like off highway Vehicles along with the influences of load, prep angle and plate thickness. In this paper, finite element simulation is performed to predict the fatigue life of full penetration weld joints used in off-highway vehicle structure. In order to simulate this in Finite element model, Transient thermal analysis is performed to simulate the weld under moving heat source conditions, which is coupled with static structural to perform fatigue analysis under strain life based conditions. Finite element analysis results are compared with literature results. In order to investigate the influences of load, prep angle (Prep - Preparation) and plate thickness, one of design of experiments (DOE) approach i.e. response surface methodology (RSM) is used to decide the number of finite element simulations. Analysis of variance (ANOVA) is performed to find the intensity of influences of load, prep angle and plate thickness on fatigue life. Using RSM optimizer, optimized level of parameters is found out to get maximum fatigue life.

2. Modeling and Finite Element Simulation

Figure 1 explains the steps involved during the finite element simulation for residual stress and fatigue assessment. As it is seen in Fig. 1, the transient thermal analysis with input heat conditions is performed to simulate the moving heat source. The temperature fields or thermal load is obtained which is coupled with Static Structural analysis as imported loading condition for fatigue assessment and the corresponding fatigue life is estimated. This finite element procedure is repeated for different levels of parameters like prep angle, load and plate thickness using design of experiments.

Doubler plate backing strip weld joint is one of the most typical weld joints used in off-highway vehicle structures as shown in Figure 2 and Figure 3. It is under the category of full penetration weld joints and has the temporary backing strip to achieve the full penetration. Addition of doubler plate in big welded structures is to achieve the required stiffness in local area instead of changing the thickness of complete structures. It also helps to design the structures with lower weight, better performance along with improved fuel efficiency. Possibility of stress concentrations, in case of doubler plate weld joint at the weld toe and root, is very high due to the geometrical discontinuities. If the weld joint parameters are not optimized properly, it leads to fatigue cracks initiation from these locations. According to the actual geometry of doubler plate backing strip weld joint structures in off-highway vehicle applications, the load conditions are chosen.

The Doubler plate weld joint consist of a constant thickness plate, a varying thickness plate (The Doubler Plate), a base plate and a backing strip. The geometry of the joint is shown in Fig. 4. Axial load is applied as shown in Fig. 4. The root gap between the plates is 0.5mm. A Computer aided design (CAD) model is shown in Fig. 5. The material used here is IS2062 E50 Steel Alloy and automatic metal arc process simulation is performed with Argon + 2% oxygen gas environment. The weld heat input conditions and the properties of the material used are shown in Tables 1 and Table 2 respectively.

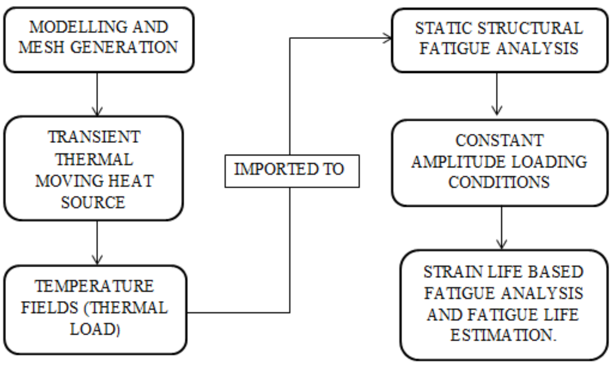

Fig. 1. Coupled Finite element analysis procedure

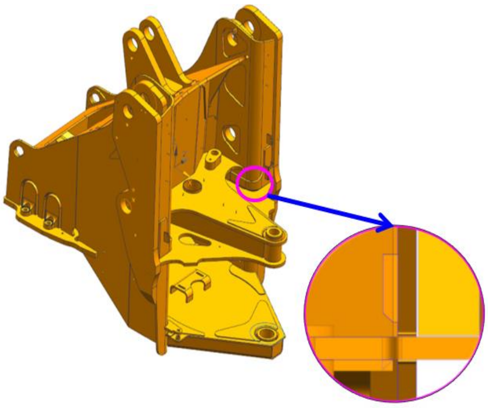

Fig. 2. Wheel Loading Shovel Front Chassis

Fig. 3. Fatigue Failure of Doubler Plate

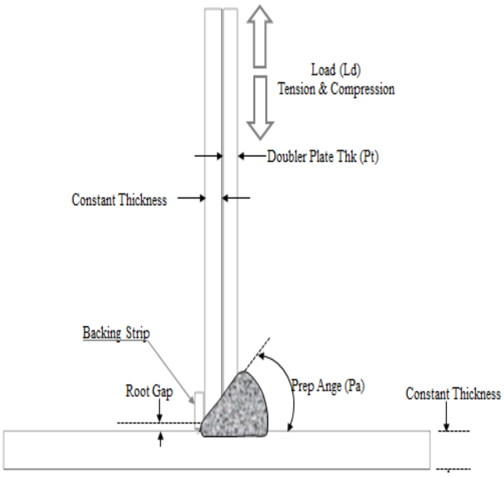

Fig. 4. Geometry of The Joint

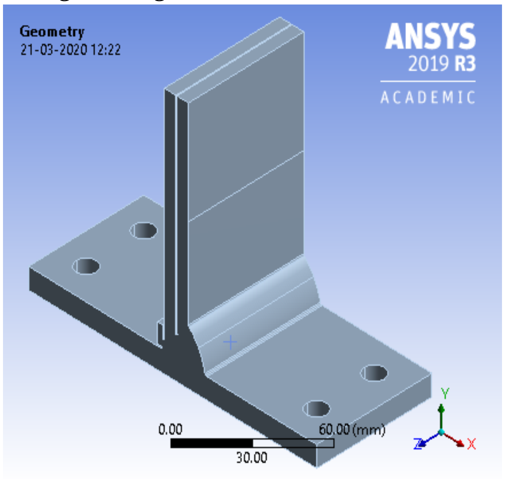

Fig. 5. CAD Model of joint

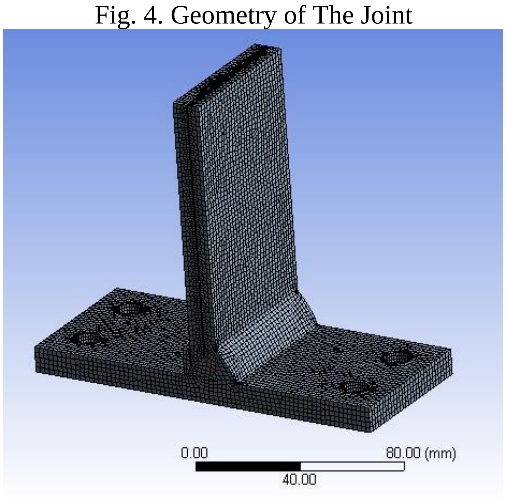

Fig. 6. Hexahedral Meshed Model

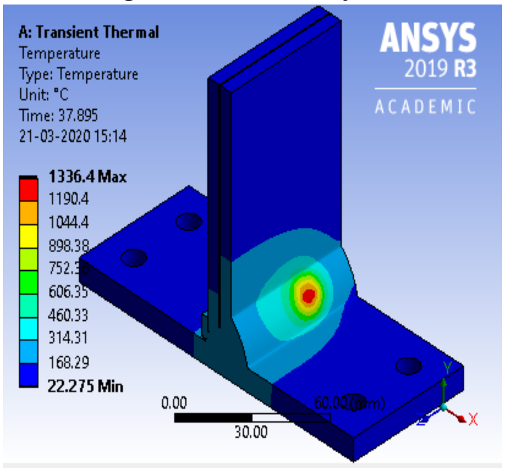

Fig. 7. Moving Heat Source at 37th Second

Table 1

Welding Specifications

S.No	Parameter	Specification
1	Arc Efficiency	75% or 0.75
2	Current in Amps	130-200
3	Voltage	17-19V
4	Speed	318-508 cm/min
5	Optimum Current	160A
6	Optimum Voltage	18V
7	Optimum Speed	381 cm/min

Table 2

Material Properties of Welded Joint

Steel Grade	Chemical Composition					Yield Strength (MPa)	Tensile Strength (MPa)
	C	Mn	S	P	Si		
E350 (IS2062)	0.20	1.55	0.040	0.040	0.45	350	490

The transient thermal analysis of the weld joint is considered. The moving heat source simulation is performed here for the welded joint. Standard FEA Package ANSYS Academic 2019 R3 has been used. The FEA model is meshed with 1.0 mm element size hexahedral mesh as shown in figure 6. The heat input is calculated by an empirical relation presented in equation 1 and the input is given in the form of heat flux [17].

$$Q = \eta AV/U \tag{1}$$

where Q=Heat Flux or Heat Input; η = Arc Efficiency; A = Current; V= Voltage; U = Speed

The convective heat transfer coefficient of argon and air is also considered since the gas metal arc welding is used here. The convective heat transfer coefficient of argon (15 W/m^2) is applied on the weld zone and the convective heat transfer coefficient of air (3.5 W/m^2) is applied on the adjacent sides of the welded joint. The welding is simulated for 60 seconds across the weld. The cooling time of the weld is also considered. The resultant thermal load body temperature is considered as an input condition for static structural fatigue life simulation. Fig. 7 shows the movement of heat source at 37th second. For the fatigue analysis of the welded joint, the imported thermal load data i.e. the temperature field is considered as input condition which is shown in Fig. 8. The strain life-based fatigue analysis performed on the welded joint under constant amplitude axial loading conditions is shown in figure 9. The mean stress correction theory employed here is Smith-Watson-Topper (SWT Theory) and the alternating stress and strain are noted down from the obtained Hysteresis Curve.Due to cyclic loading and unloading, one can obtain this curve from FEA Simulation. The theoretical hysteresis curve is shown in Fig. 10 and the obtained hysteresis curve from FEA is shown in Fig. 11.

The alternating stress from the fatigue tool hysteresis curve is noted for obtaining the number of cycles. The SN curve for the full penetration weld is BS7608 Class E Curve as per Fatigue Design and Assessment of Steel Structures BS7608:1993. From the curve, the corresponding stress vs. number of cycles is interpolated. The BS7608 SN Curve is shown in Fig. 12. Table 3 shows the validation of present model with literature results from Deshmukh [16]. The error percentage is just below 6% which shows the accuracy of the present model. Hence it is decided to carry out the further analysis with the present models for different combinations using DOE approach.

Table 3

Validation of present model with literature result

Plate Thickness (mm)	Prep angle	Load (kN)	Fatigue life (Present model)	Fatigue life [16]	Error
5	60	7.5	181137	171184	5.82%

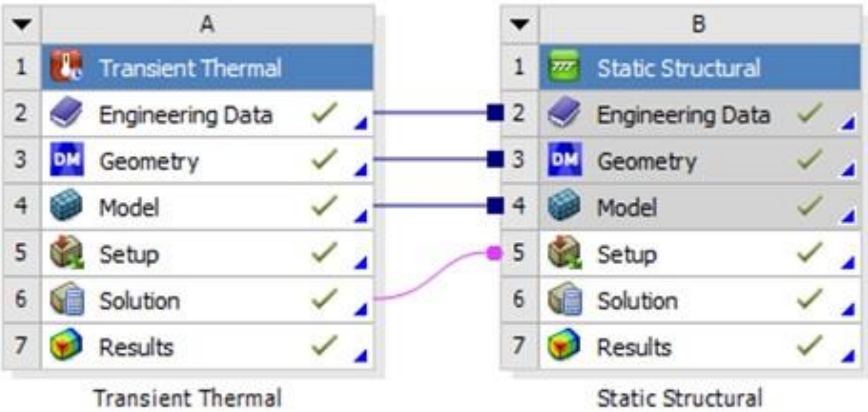

Fig. 8. Importing the thermal data for fatigue assessment

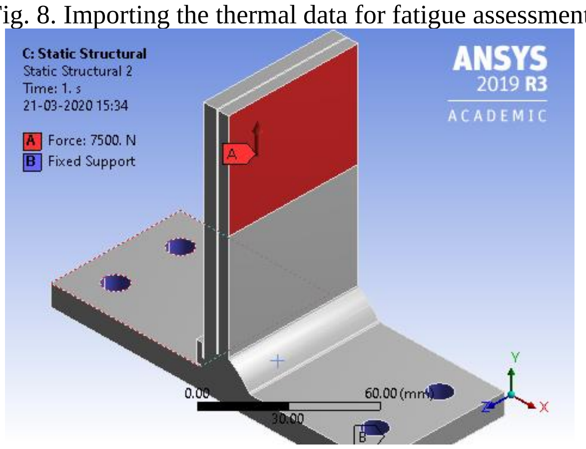

Fig. 9. Loading conditions for fatigue analysis

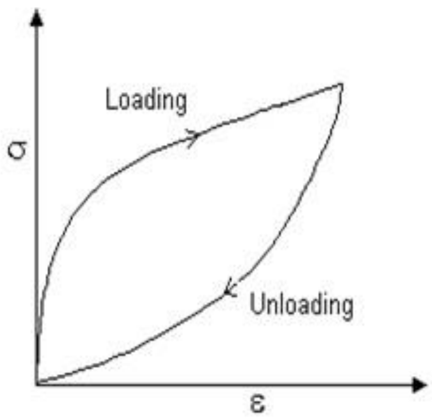

Fig. 10. Theoretical Hysteresis Curve

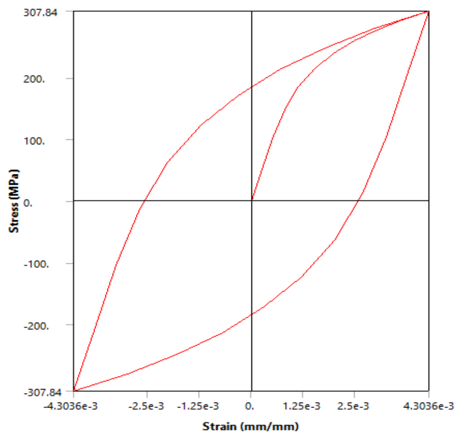

Fig. 11. Obtained Hysteresis Curve

3. Results and Discussion

3.1. Design of Experiments and Optimization

The influencing parameters in the doubler plate welded joint are plate thickness, prep angle and the applying load. Five different levels are considered in each parameter. The design of experiment (DOE) approach is used to find the different combinations of plate thickness, prep angle and the applying load to perform simulation. The Central Composite Design (CCD) of Response Surface Methodology (RSM) is used in the DOE approach as shown in Table 4. The CCD is the most popular of the many classes of RSM designs and is very efficient, providing much information on variable effects and overall error in a minimum number of required runs. It is the combination of 20 points with 6 axial and 8 corner points. Each of these process parameters is set at 5 different levels. All three factors are categorized as -2, -1, 0, 1 and 2 to have a rotatable design. A Central Composite Design in RSM is used to define the plan with 20 simulation runs as given in Fig. 13 and the values of different parameters used along with their levels are as shown in table 4.CCD is very flexible. Fig. 13 shows the CCD of RSM. Here the response parameter is the fatigue life cycles.

The DOE table is used to decide the number of finite element simulation to be performed. The sample models are created accordingly. The transient thermal and static structural fatigue analysis are performed on the sample models. From the strain life based analysis, the alternating stress and strain curve is obtained as result. The corresponding number of cycles is obtained using BS7608 Class E SN Curve and the same is noted down as shown in table 5. The analysis of variance (ANOVA) is performed to understand the influences of input parameters such as plate thickness, Load, weld prep angle on response parameter i.e. fatigue life cycle. Table 6 presents the ANOVA results. The relationship between input parameters and the response parameter is obtained using regression equation and it also helps to understand the relationships among the variables in the model, further allowing more hypotheses to be tested.

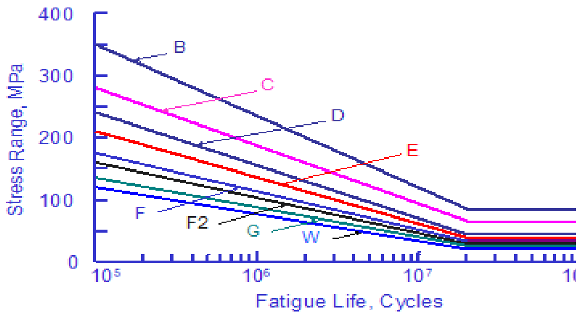

Fig. 12. BS7608 Weld SN Curves

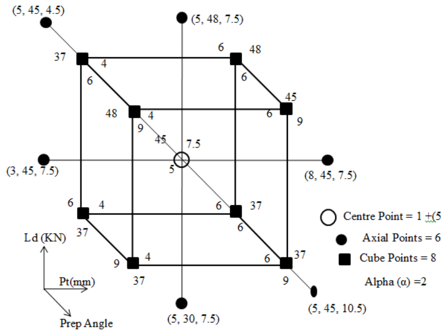

Fig. 13. Central Composite Design

Table 4

Parameter Levels

Symbol	Weld Joint Parameters	Levels				
		-2	-1	0	1	2
X1	Plate thickness (mm)	3	4	5	6	8
X2	Prep Angle (deg)	30	37	45	48	60
X3	Load (kN)	4.5	6	7.5	9	10.5

Table 5

DOE Table showing different combinations and fatigue life cycle

Standard Order	Run Order	Plate Thickness (mm)	Prep angle	Load (kN)	No of Cycles
12	1	5	45	10.5	177889
15	2	5	45	7.5	185554
19	3	5	45	7.5	185554
5	4	4	48	6	188285
11	5	5	45	4.5	190393
2	6	6	37	6	192842
13	7	5	30	7.5	188376
8	8	6	48	9	184459
7	9	4	48	9	186096
14	10	5	60	7.5	181137
18	11	5	45	7.5	185554
9	12	3	45	7.5	193107
16	13	5	45	7.5	185554
17	14	5	45	7.5	185554
10	15	8	45	7.5	173519
4	16	6	37	9	185428
1	17	4	37	6	182929
3	18	4	37	9	187222
20	19	5	45	7.5	185554
6	20	6	48	6	189596

Table 6

Analysis of Variance (ANOVA) results

Source	DF	Adj SS	Adj MS	F-Value	P-Value
Model	9	286210329	31801148	2.82	0.061
Linear	3	77863409	25954470	2.30	0.139
Plate Thickness (mm) – *A*	1	70732299	70732299	6.27	0.031
Prep angle – *B*	1	34200914	34200914	3.03	0.112
load (kN) – *C*	1	21606995	21606995	1.92	0.196
Square	3	16587125	5529042	0.49	0.697
A^2	1	13998704	13998704	1.24	0.291
B^2	1	4868243	4868243	0.43	0.526
C^2	1	3812185	3812185	0.34	0.574
2-Way Interaction	3	69058665	23019555	2.04	0.172
AB	1	35815288	35815288	3.18	0.105
AC	1	26846128	26846128	2.38	0.154
BC	1	6397249	6397249	0.57	0.469
Lack-of-Fit	10	112792783	11279278		
Pure Error	5	112792783	22558557	*	*
Total	5	0	0		
Total	19	399003112			

The MINITAB was used perform The ANOVA. AdjSS – Adjusted Some of Squares; AdjMS – Adjusted Mean Squares; F-Value Helps To Decide "Accept Or Reject" The Hypothesis. P-value helps to find the significance of results.

3.2. Regression Equation

The obtained regression equation, presented in equation 2, is a second order polynomial equation. From the regression equation, the corresponding values of plate thickness, prep angle and load are substituted for different samples and the number of cycles is tabulated in Table 7 along with the percentage of error calculation between the number of cycles calculated from Finite Element simulation and Regression equation.

$$\text{No. of Cycles} = 17523 + 28368A + 3102B + 11768C - 481A^2 - 7.8B^2 - 175C^2 - 365AB - 1221AC - 103BC \quad (2)$$

Where A = Plate Thickness; B = Prep Angle; C = Load

The regression model is tested and compared with FEA results to understand its compatibility. The calculated error in fatigue life cycles between finite element simulation and regression equation is less than 5%. This shows the legitimacy of the regression equation.

In order to find the effect of input parameters on the fatigue life cycle, the main effects plot is plotted between the influencing parameters and the fatigue life. The main effects plot is shown in Fig. 14. It is palpable from Fig. 14 that the fatigue life of the weld joint is reduced when the load is increased. There is a surge in number of cycles when the prep angle is between 45 to 48 and plate thickness between 5 to 8 mm. This shows a mixed influence in fatigue life. The corresponding contour plots are also plotted as shown in Figs. 15-17. The contour plots show the influence of the plate thickness, prep angle and load on fatigue life weld joints.

3.3. Multiple Response Prediction

The response optimization study is performed to identify the optimized combination of variables that jointly optimize a single response or multiple responses. This is useful to evaluate the effect of multiple variables on a response. Figure 18 shows the optimization plot. RSM is not just a DOE tool but an optimizer itself. In the MINITAB, an option is available to get optimization value if RSM is used as DOE tool. Fig. 18 and table 8 are provided by RSM optimizer available in MINITAB software. After getting the optimized values, again it is checked / validated with our FE model. Optimized combination of "8 mm plate thickness, 4.5 kN load, and 30 degree prep angle" is given by RSM optimizer for maximum fatigue life.

From Fig. 18, It is seen that the combination of 8 mm plate thickness, 4.5 kN load, 37 degree prep angle has better response and has a fatigue life of 197636 cycles as shown in table 8. A confirmation finite element simulation is carried out as per the optimized combination specification to evaluate fatigue life. The corresponding fatigue life cycle from confirmation simulation is 187163. Error between optimized fatigue life cycle (197636) and calculated using confirmation FE simulation life cycle (187163) is 5.29%. This shows the worthiness of optimized model.

Table 7

Error analysis between FE simulation model and regression model

Std. Order	Run Order	Plate Thickness (mm)	Prep angle	Load (kN)	No. of Cycles (from FEM)	No. of Cycles (from regression equation)	Error %
12	1	5	45	10.5	177889	180508	1.451
15	2	5	45	7.5	185554	186874	0.706
19	3	5	45	7.5	185554	186874	0.706
5	4	4	48	6	188285	189483	0.63
11	5	5	45	4.5	190393	190090	0.159
2	6	6	37	6	192842	190966	0.981
13	7	5	30	7.5	188376	188081	0.156
8	8	6	48	9	184459	177526	3.904
7	9	4	48	9	186096	187428	0.7111
14	10	5	60	7.5	181137	182156	0.555
18	11	5	45	7.5	185554	186874	0.706
9	12	3	45	7.5	193107	188999	2.173
16	13	5	45	7.5	185554	186874	0.706

17	14	5	45	7.5	185554	186874	0.706
10	15	8	45	7.5	173519	176471	1.673
4	16	6	37	9	185428	184984	0.239
1	17	4	37	6	182929	185512	1.398
3	18	4	37	9	187222	186856	0.195
20	19	5	45	7.5	185554	186874	0.725
6	20	6	48	6	189596	186907	1.438

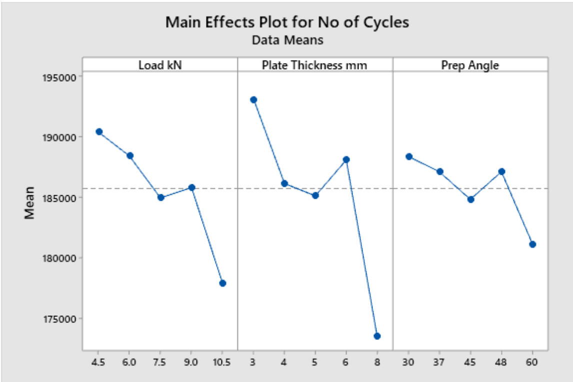

Fig. 14. Main Effects plot showing influences of different parameters on fatigue life cycle

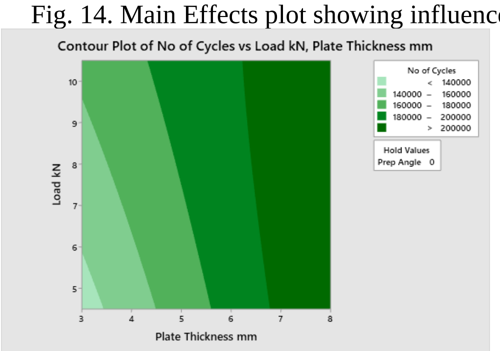

Fig. 15. Influence of load and plate thickness

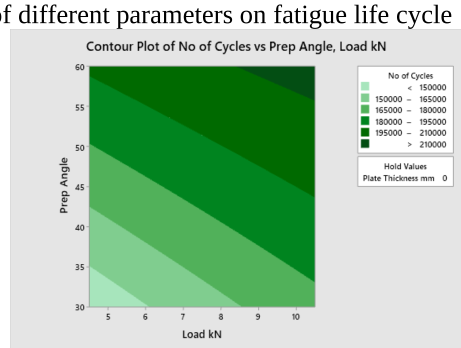

Fig. 16. Influence of load and prep angle

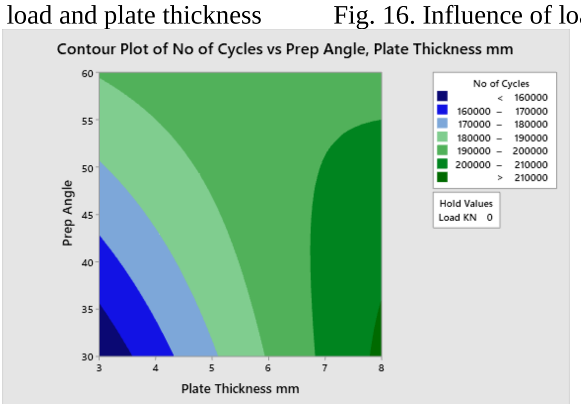

Fig. 17. Influence of prep angle and plate thickness

Table 8

Response optimization results

Variable			Setting	
Plate Thickness (mm)			8	
Prep Angle (degrees)			30	
Load (kN)			4.5	
Response	Fit	SE Fit	95% CI	95% PI
No of Cycles	203582	14015	(172355, 234810)	(171471, 235694)

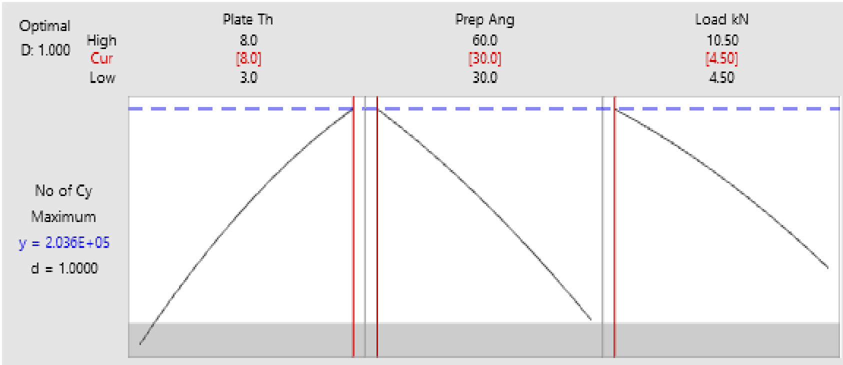

Fig. 18. Response optimization of different parameters

4. Conclusion

The finite element simulation to estimate fatigue life of doubler plate full penetration weld joint used in off-highway vehicle is carried out by incorporating the effect of residual stress. After validating the fatigue life estimated based on present model with literature result, influences of plate thickness, prep angle and load on fatigue life including the effect of residual stress are investigated using DOE / ANOVA approach. Based on the DOE approach, finite element simulations are performed for 20 different models and corresponding fatigue life is estimated. The regression equation is arrived using ANOVA and the fatigue life calculated based on regression equation is validated with finite element results. The optimization study is performed to find the optimum combination of plate thickness, prep angle and load to get maximum fatigue life. Confirmation simulation is carried out to validate optimization study. From optimization study, it is noted that the model with plate thickness of 8 mm, prep angle of 37 degree and a load of 4.5 kN gives maximum fatigue life. The developed regression equation helps to avoid repeating real experiment on machine to predict the fatigue life virtually. This will help in saving lot of time to optimize the weld joint configuration in all future design.

R E F E R E N C E S

[1] *A. Amudha, H.S. Nagaraja, H.D. Shashikala*, Finite element analysis of thermal residual stresses in SS-309Mo and Inconel-625 multilayer weld deposition on low carbon steel, International Journal of Fatigue 127 (2019),338-344.

[2] *Hafiz Waqar Ahmad , Jeong Ho Hwang, Ju Hwa Lee and Dong Ho Bae*, Welding Residual Stress Analysis and Fatigue Strength Assessment of Multi-Pass Dissimilar Material Welded Joint between Alloy 617 and 12Cr Steel, Metals MDPI(2018),8,21,1-11.

[3] *Z Samad*, N M Nor and E R I Fauzi*, Thermo-Mechanical Simulation of Temperature Distribution and Prediction of Heat-Affected Zone Size in MIG Welding Process on Aluminium Alloy EN AW 6082-T6, IOP Conf. Series: Materials Science and Engineering530 (2019) 012016, ICRAIEM 2018.

[4] *E. Ranjbarnodeh, H. Pouraliakbar, A. H. Kokabi*, Finite Element Simulation of Carbide Precipitation in Austenitic Stainless Steel 304, International Journal of Mechanics and Applications 2012, 2(6): 117-123.

[5] *Prashant Sagar, Deepak Kumar Gope and Somnath Chattopadhyaya*, Thermal analysis of TIG welded Ti-6Al-4V plates using ANSYS, IOP Conf. Series: Materials Science and Engineering 377 (2018) 012113.

[6] *A. R. Deshmukh, G. Venkatachalam, M. R. Saraf*, Fatigue Life Prediction of Doubler Plate Weld Joint Using Virtual Strain Gauge, Validation through Experiments and Its Application, International Review of Mechanical Engineering (I.RE.M.E.), Vol. 12, N. 2,196-206.

[7] *A. R. Deshmukh, G.Venkatachalam, Hemant Divekar, M. R. Saraf*, Effect Of Weld Penetration On Fatigue Life, 12th GLOBAL CONGRESS ON MANUFACTURING AND MANAGEMENT, GCMM 2014, Procedia Engineering 97 (2014) 783 – 789.

[8] *Tso-Liang Teng, Chin-Ping Fung, 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.

[9] *Tianci Li,Lele Zhang ,Chao Chang, Liang Wei,* A Uniform Gaussian distributed heat source model for analysis of residual stress field of S355 steel T welding, Advances in Engineering Software,126(2018),1-8.

[10] *X.Y.Li, T.Partanen,T.Nykänen,T.Björk,* Finite element analysis of the effect of weld geometry and load condition on fatigue strength of lap joint, International Journal of Pressure Vessels and Piping 78 (2001) 391-397.

[11] *Nazrul Islam,Tasnim Hassan*, Influence Of Initial And Welding Residual Stresses On Low Cycle Fatigue And Ratcheting Response Simulations Of Elbows , Proceedings Of The ASME 2017 Pressure Vessels And Piping Conference PVP2017 July 16-20, 2017, Waikoloa, Hawaii, USA .

[12] *Mato Peric, Ivica Garašic´, Sandro Nižetic and Hrvoje Dedic´-Jandrek*, Numerical Analysis of Longitudinal Residual Stresses and Deflections in a T-joint Welded Structure Using a Local Preheating Technique,Energies,MDPI,2018.

[13] *H.M.E. Ramos, S.M.O. Tavares, P.M.S.T. de Castro*, Numerical modelling of welded T-joint configurations using SYSWELD, Science and Technology of Materials 30 (2018) 6–15.

[14] *Dongpo Wang, Hai Zhang, Baoming Gong, Caiyan De*ng, Residual stress effects on fatigue behaviour of welded T-joint: A finite fracture mechanics approach, Materials and Design 91 (2016),211-217.

[15] *Chuang Cui, Qinghua Zhang, Yi Bao, Yizhi Bu, Ying Luo*, Fatigue life evaluation of welded joints in steel bridge considering residual stress, Journal of Constructional Steel Research 153 (2019) 509–518.

[16] *Atul.R.Deshmukh*, Virtual Fatigue Life Prediction of Full Penetration Weld Joints in Off-Highway Vehicles, PhD Thesis, 2019, Vellore Institute of Technology, Vellore, India.

[17] *Gurinder Singh Brar, Gurdeep Singh,* FEA of Residual Stresses in Butt Welded Type Low Carbon Steel Using MMAW Technique, Applied Mechanics and Materials Vols. 110-116, 2686-2692.

[18] *Atul R. Deshmukh , Gopalan Venkatachalam ,M.R. Saraf*, Fatigue life prediction of joggle weld joint using virtual strain gauge and its validation through experiments, Mechanics & Industry 19, 604 (2018).