Simulation and Modeling of the Effect of Welding Process Parameters of Inconel 625

Antony Solomon.S
Department of Mechanical Engineering
Saranathan College of Engineering, Trichy-12.

Deepak. R
Department of Mechanical Engineering
Saranathan College of Engineering, Trichy-12.

Aravinthan. G
Department of Mechanical Engineering
Saranathan College of Engineering, Trichy-12.

Kishore Kumar. M
Department of Mechanical Engineering
Saranathan College of Engineering, Trichy-12.

Anantha Padmanaban. M. R
Department of Mechanical Engineering
Saranathan College of Engineering,
Trichy-12.

Abstract **— This paper examines effect of welding parameters for Inconel 625 alloy such as stress and temperature distribution. Inconel 625 alloy welding is widely used in space vehicles, marine applications which involves high cost of welding, therefore it is important to choose welding parameter wisely. FEA simulation data requirements are met by conducting design of experiments (DOE) for three levels of welding current(100-120 A), voltage(10-14 V) and speed(75 – 150 mm/min) are modeled and heat flow is obtained which is fed as input for analysis. Coupled field analysis of transient thermal and structural analysis is done on ANSYS 15.0 and results were tabulated and fed into historical data design in design expert software. Equation was obtained from regression model and is verified.**

Keywords: Welding, Welding Simulation, ANSYS, Regression Equation, Temperature distribution, Stress, DOE.

I INTRODUCTION

Welding technology is one of the main joining methods and widely used in industries to assemble various products such as ships, automobiles, trains and bridges. For instance, the assembly process in shipbuilding essentially involves the joining of large blocks. Typically, these blocks are all-welded, thin-plate structures. During the fabrication of these blocks, distortions occur due to a variety of causes, such as cutting and welding. It has critical effects on the quality, reliability and life of products as well as production cost, efficiency and response speed to market. Metallic materials are widely used in engineering; there are more demands in manufacturing industry for advanced welding technology. In order to research and develop new welding processes with high quality, high efficiency, low cost and environment-friendly features and to ensure the safety and reliability of weld structures, it is essential to conduct modeling and simulation of welding physical phenomena and process mechanisms. [1-2]

So the simulation of welding in any FEA package is to predict stress, distortion produced on welding plates. It will help to select optimal process parameter for welding selected material. It is also used to find temperature distribution during welding over the plate .It helps to predict heat affected zone (HAZ). A regression equation is used in stats to find out what relationship if any, exists between sets of data. Design of experiments is initially done to find various feasible combinations and is fed to the HISTORICAL DATA DESIGN. The result from the various analysis models for load conditions selected from historical data design are fed as input for obtaining regression equations and are verified.

II DESIGN OF EXPERIMENT

Design of experiment is modeled based reference journal and machine capability .DOE is shown in below

TABLE I : DESIGN OF EXPERIMENT

Parameters	Symbol (unit)	Level 1	Level 2	Level 3
Welding current	I (Amperes)	100	110	120
Welding voltage	V(Volts)	10	12	14
Welding Speed	v (mm/min)	75	100	150

III MODELING AND SIMULATION OF WELDING

A) Modeling of weld plates

Design of weld joint is done according to ASME standards for selected size of inconel 625 plate[3-4]. Dimensions of the Inconel welded plate shown in Fig 1.It is modeled in Creo 3.0 parameteric software and imported in ANSYS in IGES format shown in Fig 2.

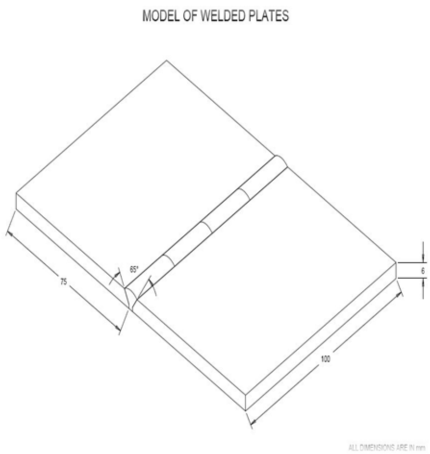


Fig .1 Model of welded plate

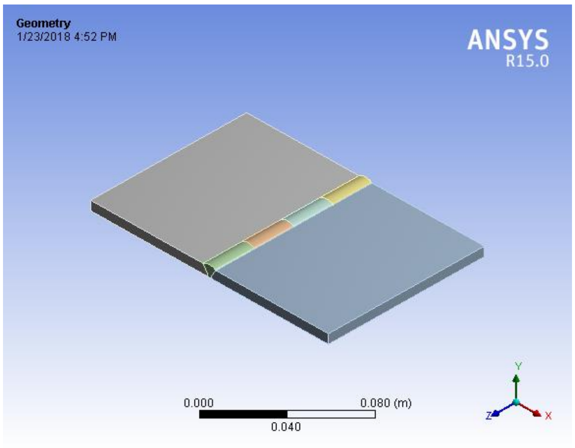


Fig .2 Creo model imported in ANSYS

The dimension of the plate as follows

Length : 100 m
Breadth : 75 mm
Thickness : 6 mm
If the plate thickness is 6mm, then
Joint type : V joint
Root opening : 3.2 mm
Included angle : 60-75 ^{o}

Chemical composition of Inconel 625 alloy

TABLE. II Chemical Composition, %

Cr	Ni	Mo	Co + Nb	Ta	Al	Ti	C	Fe	Mn	Si	P	S
20.00-30.00	Remainder	8.0-10.0	1.0 max	3.15-4.15	.40 max	.40 max	.10 max	5.0 max	.50 max	.50 max	.015 max	.015 max

B) FEA Simulation of welding

1) Material properties

a) Mechanical properties

Density : 8440 kg/m^3
Ultimate tensile stress : 1103 MPa
Co efficient of thermal expansion : 1.28E-05/C at 93ºC

Tensile Yield Strength : 758 MPa
Compressive Yield Strength : 758 MPa

b) Thermal properties

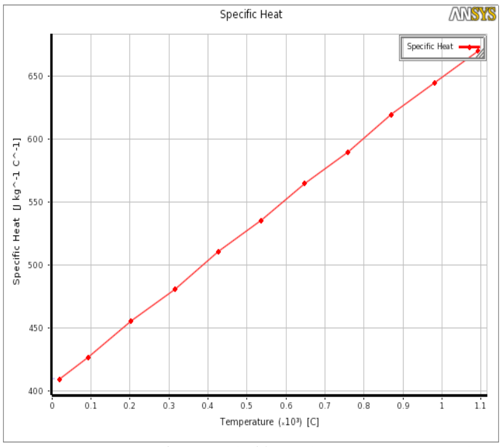


Fig 3. Specific heat (J / kg ◦C) vs Temperature (◦C)

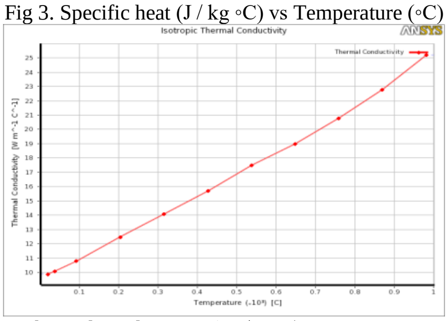


Fig. 4. Thermal conductivity (W /m ◦C) vs. Temperature (◦C)

2)Heat input calculation

Heat flow through the electrode is in proportion with voltage and current, with a transmissibility loss in transformer. It is the product of voltage , current and efficiency of transformer.

$$Q = I \times V \times \eta$$

Where,

Q = Heat flow (Watt)

I = Welding current (Amperes) ,
V = Welding voltage (Volts)
η = Efficiency of the welding transformer (Percentage)

Heat input calculation is done for selected parameters are shown in table.

TABLE. III CALCULATION OF HEAT FLOW

Assumption :Efficiency of the welding transformer η = 70 %

Ex.No	I(Amperes)	V(Volts)	V(mm/min)	Heat flow (Q)=I.V. η (W)
1	100	10	75	700
2	100	12	100	840
3	100	14	150	980
4	110	10	100	770
5	110	12	150	924
6	110	14	75	1078
7	120	10	150	840
8	120	12	75	1008
9	120	14	100	1176
10	110	10	75	770
11	110	12	100	924
12	110	14	150	1078
13	120	10	75	840
14	120	12	100	1008
15	120	14	150	1176

3)Meshing

Meshing is defined as the process of dividing the whole component into number of elements so that whenever the load is applied on the component it distributes the load uniformly called as meshing.

No. of nodes	:	36948
Elements	:	6972
Element nature	:	Fine
Minimum edge length	:	1.6 mm

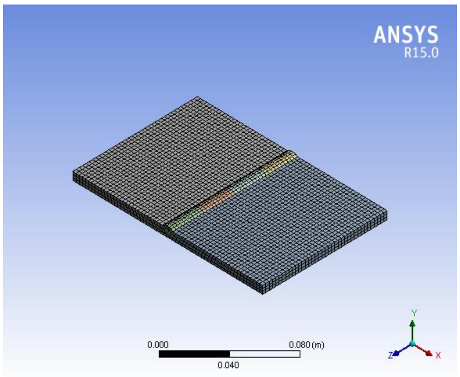


Fig.5 Meshing

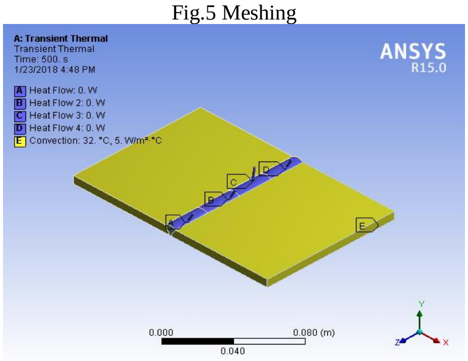


Fig 6 Transient thermal boundary conditions

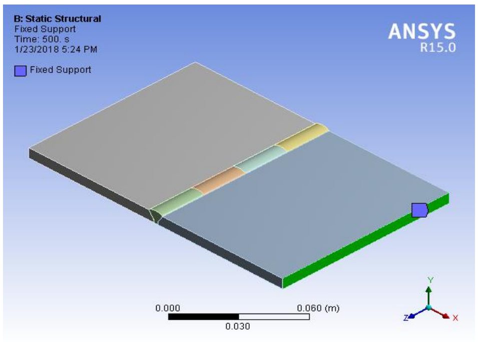
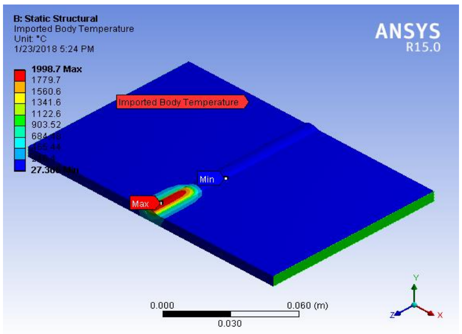


Fig .7 Fixed support



4) Boundary conditions

Simulation of welding is a coupled field analysis which means the combination of both transient and structural . Transient thermal analysis is loaded as boundary conditions for structural analysis.

a) *Transient thermal*

Heat flow shown in table 2 is applied in Thermal analysis shown in Fig .6 with respect to welding speed.

b) *Structural analysis*

Temperature distribution obtained from the transient thermal analysis is applied as load to the structural analysis in addition a fixed support is applied to adjacent sides of the plate as shown in Fig 7 and 8.

5) *Solution*

Transient thermal condition is solved for temperature distribution along the plate against time. This result is the then coupled to Static structural to determine the von mises stress distribution on the plate and total deformation were obtained.

HISTORICAL DATA DESIGN

Response Surface Models, a variation of the simple linear regression, with the incorporation of the second order effects of non-linear relationships, is a popular optimization technique to determine the best possible combinations of variables to determine a specific response to a phenomenon. A regression equation is used in stats to find out what relationship if any, exists between sets of data. Design of experiments is initially done to find various feasible combinations and is fed to the historical data design. The above table is fed into design expert software on historic data design under response surface methodology. Regression equations are obtained for stress and maximum temperature. Those equations are shown below.

A) ***Max temperature*** (Maximum temperature developed during welding)

$$T= 1157.01228 + 21.63419 * \mathbf{I} - 190.87048 * \mathbf{V} - 16.4532 * \mathbf{v} + 1.74460 * \mathbf{I} * \mathbf{V} - 0.024745 * \mathbf{I} * \mathbf{v} + 0.043620 * \mathbf{V} * \mathbf{v} - 0.1206 * \mathbf{I}^2 + 4.62762 * \mathbf{V}^2 + 0.072224 * \mathbf{v}^2$$

B) ***Stress*** (Stress developed along th length of the plate during welding)

$$Stress = 1271.03515 - 2.60674 * I - 238.35639 * V + 10.06715 * v + 1.51696 * I * V - 0.097258 * I * v - 0.47832 * V * v - 0.00519379E * I2 + 6.43346 * V2 + 0.013218 * v2$$

where,

I = Welding current (A)
V = Welding voltage (V)
v = Welding speed (mm/min)

V VALIDATION

Valitation is the process of checking the regression equation whether valid or not. It is done perform analysis for a specific parameter s and get the predicted value without analysis directly from regression equation. Selected parameter is not inclued Historical data design but near range of DOE level

TABLE . IV SELECTED INPUT PARAMETER FOR VALIDATING

S. No	Welding Current (A)	Welding Voltage (V)	Welding speed (v)
1	125	12.2	100

TABLE. V VALIDATION

Description	Actual	Predicted	Deviation(%)=[(Actual – Predicted)/Actual]* 100
Maximum temperature	1817	1818	0
Stress	538	566	5.2

Since the deviation is very small deviation, the mathematical model is valid. It is only valid between the range of respective parameters in DOE.

VI RESULTS & GRAPHS

The stress and temperature distribution graphs for the selected parameter for validation analysis shown below

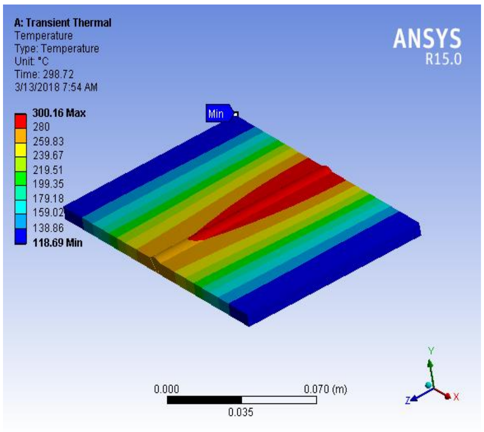


Fig .9 Temperature(°C) vs Distance (m)

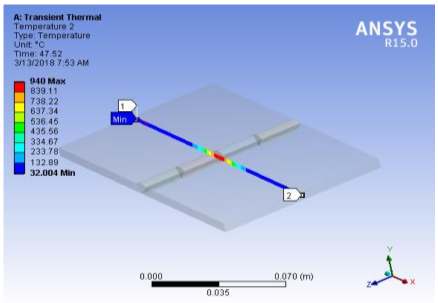
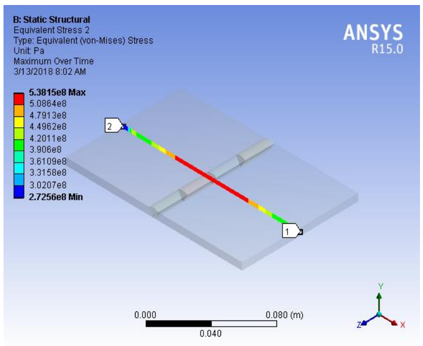


Fig .10 Stress (Pa) vs Distance((m)



Fi g .11 Temperature distribution

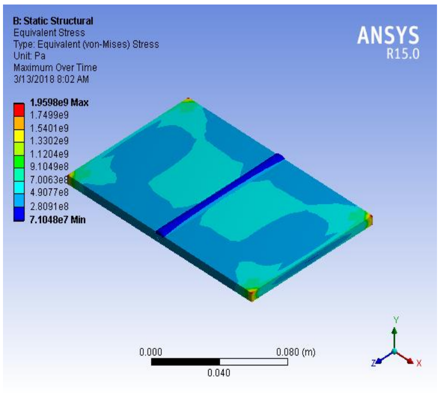


Fig . 12 Temperature distribution along length of the plate

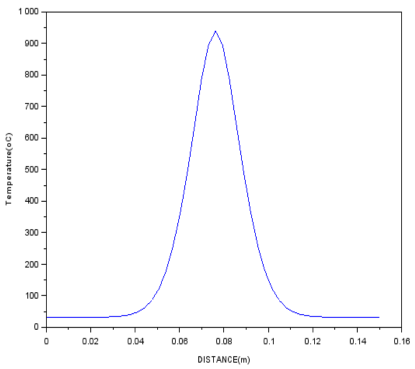


Fig .13 Stress distribution

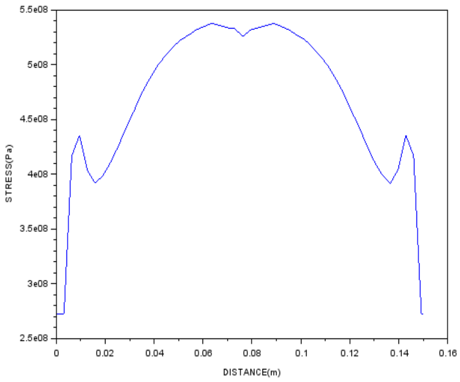


Fig . 14 Stress distribution along length of the plate

VII CONCLUSION

Simulation of welding is the process of modeling a welding operation in any FEA packages like ABAQUS, ANSYS and getting temperature, stress distribution and total deformation results. It is a coupled field analysis which means the combination of transient thermal and structural analysis. First the model is designed in CAD software and then imported in ANSYS. Inconel 625 alloy material properties imported and calculated heat flow is given as input boundary condition with respect to welding speed.Temperture distribution over the plate and particular length are obtained. Now that temperature distribution is given as input to structural analysis in addition fixed support is given to the adjacent sides of the plate. The maximum stress acting on the center of weld area is tabulated. Above analysis is done for each experimental trial as per Design of Experiment. Finally all the trails results are tabulated and enter into the Historical data design under Response Surface Methodology in Design Expert software. The mathematical model for conducted analysis obtained. It is validated by randomly selected parameter and compared the actual by Ansys and predicted by mathematical model. Since the deviation is less than 10 % the mathematical model is valid. The maximum temperature majorly depends on welding current and stress developed on the weld area is depends on welding speed was found from this work.

VII REFERENCES

[1] Deng D, Murakawa H, Liang W. Numerical simulation of welding distortion in large structures. Computer methods in applied mechanics and engineering. 2007 Sep 15;196(45-48):4613-27.

[2] Wu, Chuan-Song, Michael Rethmeier, and Christopher Schwenk. "Simulation of welding." (2011): 77-78.

[3] Welding Handbook, Ninth Edition, Volume 1, Welding Science and Technology, Chapter 5, Design for Welding, pg. 157-238, American Welding Society, 2001.

[4] ASM Handbook, Volume 6, Welding, Brazing and Soldering, Welding of Nickel Alloys, pg. 740-751, ASM International,1993.