Simulation of Thermal Stress of SS316 using ANSYS

Harinadh Vemanaboina[1] G.Edison[2] Suresh Akella[3]
[1]Research Scholar [2,3]Professor
[1,2]SMEC, VIT University, Vellore, Tamilnadu, India [3]Sreyas Institute of Engineering & Technology, India

Abstract— Welding is widely used in all the fabrication processes for the development of complex structural components. The weld distortion is one of the major constraints which cannot be completely avoided irrespective of material type and thickness. 3D transient thermal finite element model is established to measure the thermal stress and weldments. The temperature stress modeling is one of the complex processes which utilize the weld parameters and material properties at higher temperatures. A Suitable heat flux is to be given as input for the developed model to analyze welding process. The temperature distribution and stress analysis have been carried out with developed model by using the temperature dependent material properties of SS316 using ANSYS.

Key words: Welding, Thermal Stresses

I. INTRODUCTION

Welding is a process used in the fabrication of various steel structures for applications from thin sections to thicker sections in various applications like pressure vessels, chemical plants and nuclear reactors. The main problem associated with welding is the presence of residual stresses and deformations developed within the sections which may cause the failure at the later stages by Masubuchi [7]. During the welding process, the material exposed to heat flux causes a phase change of the metal during melting. The temperature distribution is non-uniform like fusion zone with molten metal, heat affected zone and the base metal zone.

The present study is focused on the understanding of the heat flux mechanism with heat source used for

TIG weld process with developed constant heat source model applied to the stainless steel, SS316 material by using the temperature dependent properties and finite element approach. The temperature distribution is estimated which can further give the estimation of the weld deformations and stresses.

II. FORMATION & SIMULATION OF WELDING PROCESS

The heat energy equations are referenced in many including Frewin [5]. For an isotropic, conductive material with equal coefficient of conductivity k_x, k_y, k_z (W/mK) in all three chosen orthogonal coordinates. Equation (1) gives the heat energy in the weld area with temperature, T (K) obtained both in spatial, x, y, z (m) and temporal, t (sec), terms. Q (W/m^3 or J/m^{3}s) is the net heat from the input and the losses in the form of convection & radiation. The boundary conditions given are $T_0(x, y, z, 0)$ throughout the body at time zero or at the starting of the weld, this is an essential boundary condition. In addition, the natural boundary conditions have to apply consisting of normal conduction $\boldsymbol{K_n \frac{\delta T}{\delta n}}$ heat flux q, convection h (T-T$_0$), and the radiation term, $\sigma\varepsilon\,(T^4 - T_0^{\;4})$

$$kx\frac{\delta^2 T}{\delta^2 x} + ky\frac{\delta^2 T}{\delta^2 y} + kz\frac{\delta^2 T}{\delta^2 z} + Q = \rho c\left[\frac{\delta T}{\delta t} - v\frac{\delta T}{\delta x}\right] \qquad (1)$$

Together, the boundary conditions are summed up as:

$$K_n - q + h(T - T0) + \sigma\varepsilon\,(T^4 - T0^4) = 0 \qquad (2)$$

When symmetric boundary and insulation boundaries are considered as adiabatic, with no heat flowing through the surface, they are obtained by making convection zero, and conduction zero from the surface. Where, K_n is the thermal conductivity normal to the surface in W/mK, h is the convective heat transfer coefficient in W/m^{2}K, ε is emissivity of surface radiating, σ is the Stefan Boltzmann's constant, which 5.67*10^{-8}, W/m^{2}K^4. When it is difficult to use radiation boundary condition, it is combined with convective heat flux by using a modified coefficient, h_r, for hot rolled steel plates with an error of about 5% is,

$$h_r = 2.4 * 10^{-3} \in T^{1.61} \qquad (3)$$

Radiation inclusion will increase solution time by about three times and hence combined with convection.

A. Finite Element for Simulation

The heat equations (1) can be represented in tensor form so the elemental transient heat equation is obtained and later summed to get the system equation which is analysed with time.

$$[K(T)]\{T\} + [C(T)]\{T\} = \{Q(T)\} \qquad (4)$$

Where K is a temperature dependent conductivity matrix. C is the temperature dependent capacitance matrix based on specific heat its product with the rate of temperature gives heat. The above equation can be solved numerically, with standard FEM models with Crank-Nicholson or Euler time integration models. An initial temperature T_i is assumed K, C and

Q are calculated at that temperature and the next temperature *T* at i+1 is obtained. Again *K*, *C* & *Q* are calculated and temperature at next temperature interval is calculated.

B. Finite Element Model

The finite element model of dimensions 40mm X 150 mm X 5 mm is used. The AISI 316 austenitic stainless steel material is considered for simulations to be carried out. The convection is applied on all the surface of the plate except on the heat applied area. In the present study AISI type 316 stainless steel is used as it is having many advantages such as low thermal conductivity, the high resistance to corrosion and high stability at elevated temperatures. Thus, SS316 material is widely used in numerous industries, like nuclear industry, chemical plants, aeronautical and specialized pipe industry.

III. THERMAL ANALYSIS

In the present work Finite Element Analysis of single-pass butt-welding has been carried out with constant heat flux, the load is applied for first 10 sec and allowed to cool to ambient temperature for 1000 sec. For this, a simple Butt-joint welding whose welding parameters are consistent to those of Friedman's model with heat input Q = 1200 W is considered and has been simulated using ANSYS. The element can also compensate for mass transport heat flow from a constant velocity field. In this analysis, element SOLID70 is replaced with by a three-dimensional (3-D) structural element SOLID45. The geometry and meshed model with tetrahedral shape with volume mesh of size 0.02 were shown in Fig 1(a) , Fig 1(b).and Fig 1(c).

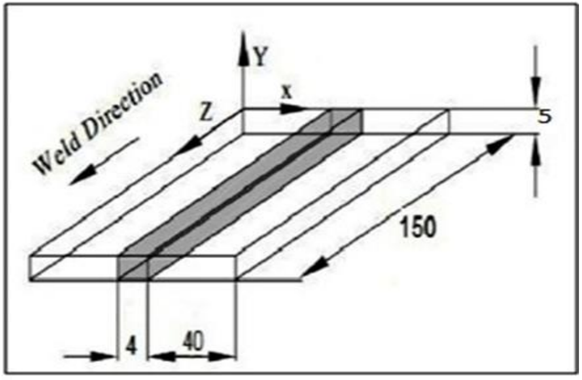

Fig. 1(a): Model Dimensions for Simulation

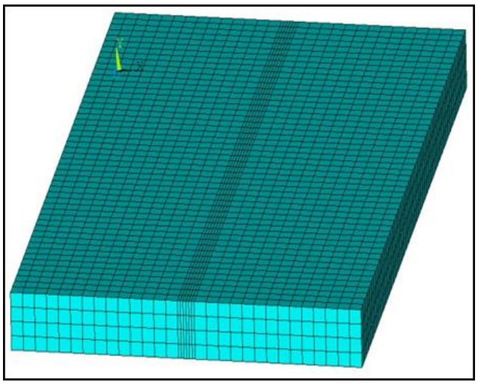

Fig. 1(b): Mesh Model used for ANSYS

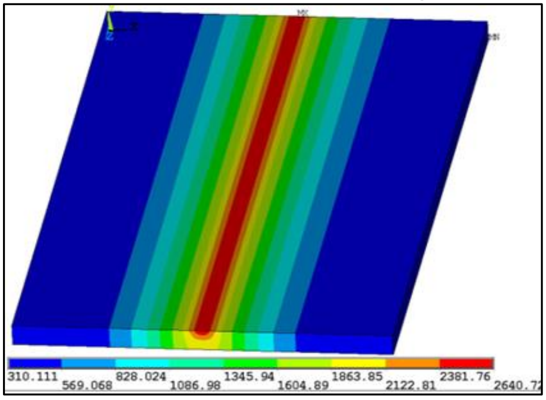

Fig. 1(c): Temperature distribution of the weldment

The temperature distribution was evaluated at various zones i.e. fusion zone, heat affected zone and base plate. The temperature distributions of the weldments are shown in Figs (1a) to (3). Temperature measurement studies in various zones of the model and to understand the distribution. Fig 3. Shows the temperature versus time distribution graphs in the three regions of Fusion zones, Heat Affected zones & Base plate. The temperature is found to vary from 303^0K in the base plate and up to 2700^0K in the fusion zone with the applied heat input parameters in the developed model. Fig (2) shows the temperature distribution in the transverse direction on the surface of the weldment.

Temperature (K)	Density (kg/m^3)	Specific heat(J/kg K)	Thermal conductivity (W/m K)
273	8038.7	456.28	13.29
293	8030.47	464.73	13.63
373	7997.02	494.23	14.99
473	7954.03	522.74	16.62
573	7909.76	543.92	18.19
673	7864.18	559.87	19.72
773	7817.31	572.69	21.26
873	7769.13	584.49	22.81
973	7719.66	597.38	24.42

1073	7668.9	613.45	24.09
1173	7616.83	634.82	27.86
1273	7563.47	663.58	29.76
1373	7508.81	701.85	31.81
1473	7452.85	751.72	34.03
1573	7395.6	815.3	36.46
1643	7354.75	869.09	38.29

Table 1: Thermal temperature dependent properties

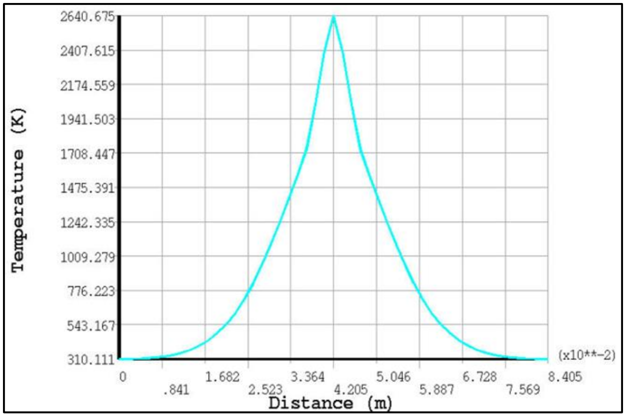

Fig. 2: Transverse Temperature distribution of the weldment

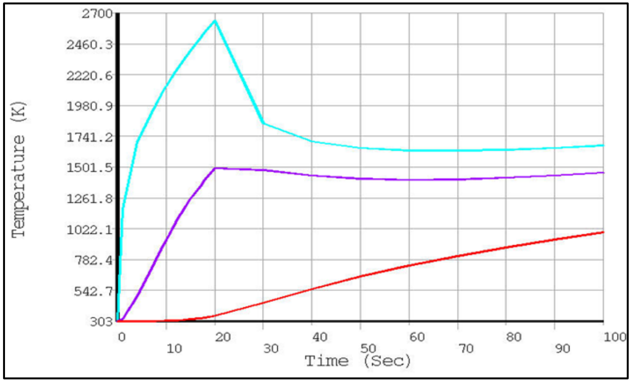

Fig. 3: Temperature distribution of the weldment at fusion zone, Heat affected zone

IV. CONCLUSION

FEM simulation by adapting a constant heat flux analysis has been carried out on SS 316 steel material for the transient thermal temperature analysis. The temperature field at the weld zone was found higher at the given constant heat flux input when compared with the heat affected zone and base plate regions. This prediction of analysis with reference to the estimation of stress can be useful to predict the weld residual stress status with the heat input parameters which can be used for the experimental application.

REFERENCES

[1] Akbari Mousavi. S.A.A, R. Miresmaeili., 2008. Experimental and numerical analyses of residual stress distributions in TIG welding process for 304L stainless steel, Journal of materials processing technology, 208, 383–394.

[2] Amudala Nata Sekhar Babu, et.al. 2012. Finite Element Simulation of HybridWelding Process for Welding 304 Austenitic Stainless Steel Plate, IJRET Vol 1, Issue 3, 101-410.

[3] Dean Deng, Hidekazu Murakawa. 2008. "Prediction of welding distortion and residual stress in a thin plate butt-welded joint" Computational Material Science 3,353-365.

[4] Dike.J.J, A.R Ortega and C.H. Cadden, .1999. Finite element modelling and validation of residual stresses in 304L girth welds, in Trends in Welding Research, 5th International Conference, ASM International, Materials Park, Ohio, pp.961-966.

[5] Frewin m r, Scott d a, "finite element model of pulsed laser welding", welding j, 1999, 15s – 22s

[6] Goldak.J.A, B. Chakravarti and M. J. Bibby. 1984. A new finite element model for welding heat sources, Metallurgical Transactions B, 15, 229-305.

[7] Masubuchi.K, .1980.Analysis of Welded Structures. Oxford, Pergamon Press. Mollicone.P, et al,.2006. Simple thermo-elastic-plastic models for welding distortion simulation. Journal of Material Processing Technology,176, 77 – 86.

[8] Shan.X, C.M. Davies, T. Wangsdan , N.P. O'Dowd, K.M. Nikbin,.2009. Thermo-mechanical modelling of a single-bead-on-plate element Method, International Journal of Pressure Vessels and Piping 86, 110–121

[9] Taljat.B, B. Radhakrishnan, T. Zacharia, 1998. Numerical analysis of GTA welding process with emphasis on post-solidification phase Transformation effects on residual stresses, Materials Science and Engineering A246 45–54.