FEA Analysis on Weld Structure

Prof. Vishawanath Kanal,

Department of Mechanical Engineering BLDEA CET Bijapur/Karnataka/India

Abstract:- **In this work, the quality of a butt welded joint is analyzed using ANSYS Finite Element Analysis Software. It is modeled as a three dimensional transient heat conduction problem. The process of welding is simulated as a moving heat source in the welding zone. The temperature distribution for different heat flux and weld speed are obtained from the Thermal analysis. The result of the Thermal analysis is given as input to the structural analysis and the coupled field analysis is carried out. The effect of input parameters such as heat flux and weld speed is analyzed.**

It has been found from the investigations that the weld speed and heat flux greatly influence the temperature distribution and residual stress developed in the welded area which in turn improve the quality of weld. Thus, the optimum parameter for quality welding has been found as minimum heat flux and maximum weld speed.

Keywords: welding, welding joint, plate, residual stre sses, stress analysis, finite element method.

I. INTRODUCTION

The increased globalization of industry is causing acceleration in the pace of product change. Shorter product development time with Excellency in functionality, quality, cost competitiveness and aesthetics is the order of the day. This trend is forcing the Engineers and Engineering managers to respond with products that have increasingly lower costs, better quality and shorter development times.

Welding is a process of joining similar or dissimilar metals by the application of heat with or without the application of pressure and addition of filler material. The result is a continuity of homogenous material of the composition and characteristics of two parts which are being joined together. The application of welding are so varied and extensive that it would be no exaggeration to say that there is no metal industry and there is no branch of engineering that does not make use of welding in one form or another. In fact, the future of any new metal may depend on how far it would lend itself to fabrication by welding**.**

II. THERMAL AND STRUCTURAL ANALYSIS OF WELDING

The type of welding under investigation is Metal Arc Welding butt joint of Mild Steel. The specimen, which is to be welded, is having a thickness of 6mm for studies. Two weld plates each of 5cm length and 2.5cm breadth is to be joined by butt joint with the help of Metal Arc Welding. The diameter of the electrode considered is 6mm

2.1 Governing Equations

The governing differential equation for the above conditions will have the following Heat Transfer mechanisms. 3-D transient heat conduction within the solid Convection heat loss from the surfaces.

The transient heat conduction equation incorporating the heat flux, surface convection to be solved is given by

$$\rho C_p \frac{\partial T}{\partial \tau} = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\partial T}{\partial z}\right) + q - 2h(T - T_{\infty})$$

Where, ρ is the density, Cp is the specific heat, k is the thermal conductivity, q is the surface heat flux (power source) intensity, T is the temperature, τ is the time, h is the convective heat transfer coefficient, and T_{∞} is the surrounding temperature.

2.2 Boundary Conditions for thermal analysis

The following are the Boundary conditions employed during the thermal analysis. At time τ =0, the entire solid is assumed to be at an uniform temperature of 30° C. The moving heat source is modeled by applying and deleting heat flux over the corresponding areas with respect to the welding speed.

At time τ >0, over the entire surface of the solid, convection heat loss is applied by specifying the constant convective heat transfer co-efficient (h) and surrounding temperature (T_{∞}).

2.3 Boundary conditions for structural analysis

The following are the Boundary conditions employed during the structural analysis. At time τ =0, displacement along the directions of plates (Ux, Uy and Uz) are assumed to be zero.

At time τ >0, displacements of left and right edges of the plates and bottom faces are assumed to be zero.

III. METHODOLOGY

Thermal and Structural analysis of the welding structure is carried out using commercial Finite Element Analysis (FEA) software **ANSYS 10**. The procedures for the thermal and structural analysis are discussed below

Modeling of weld joint (25mmX25mmX6mm)

Meshing and load application of thermal loads (Nature convection)

Analysis of the same thing with forced convection

Comparison of both convection results and forced convection

Analysis of the results under thickness effect.

3.1 Model Geometry

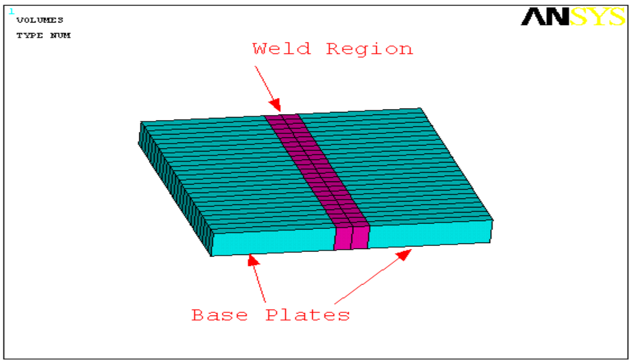

Fig 3.1 – 3D model of the weld plate.

The Fig.3.1 shows geometry of the weld problem. The middle 6mm thickness region with 6mm height of weld is considered for the analysis. Base plates are of 25mm width by 50mm length. The region is split in to 25 parts to simulate the weld process in 25 seconds for single run.

IV. RESULTS AND DISCUSSION

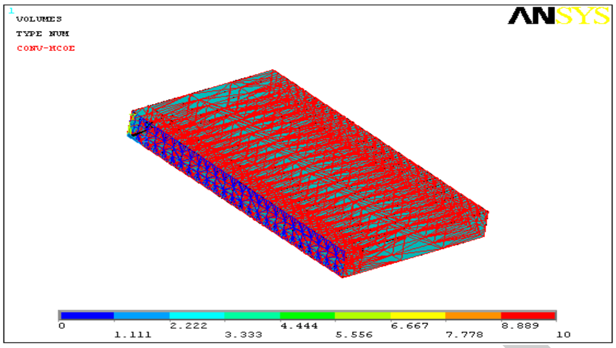

Fig.4 Boundary conditions on the half symmetrical geometry

A uniform temperature of 30°C is applied over the entire solid region as the initial condition.. Convective heat transfer is applied to all free surfaces with heat transfer co-efficient of 20 w/m^2 k. For convection, the surrounding temperature of 30˚C is specified with the convective heat transfer. Heat flux values are applied to different areas. Load steps are executed from 1 to 25to solve the problem.

Thermal structural analysis is carried out with different heat flux on the weld geometry.

Case 1 : Heat Input : 4e7w/m^2

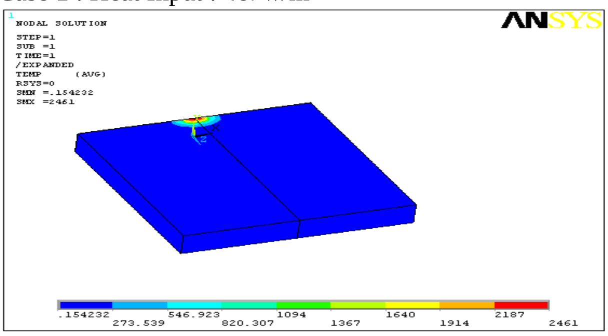

Fig 4.1 : Temperature distribution at start of weld

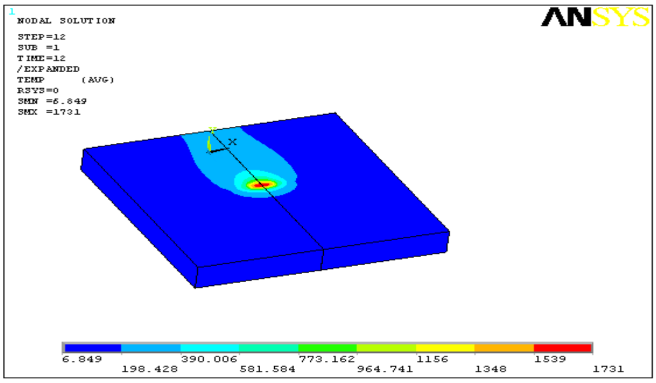

Fig 4.2 Temperature distribution after 12 sec of start of weld

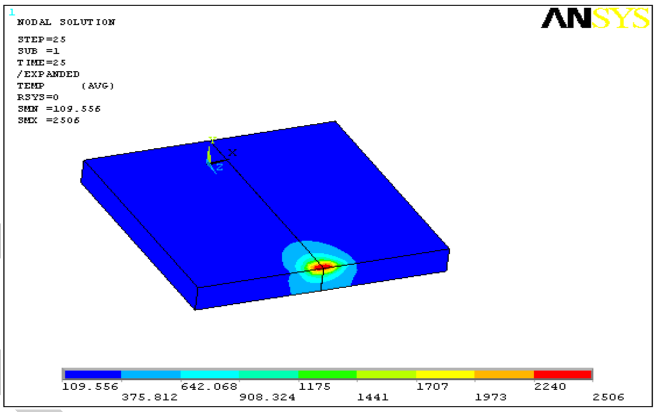

Fig 4.3 Temperature distribution after 25 sec of start of weld

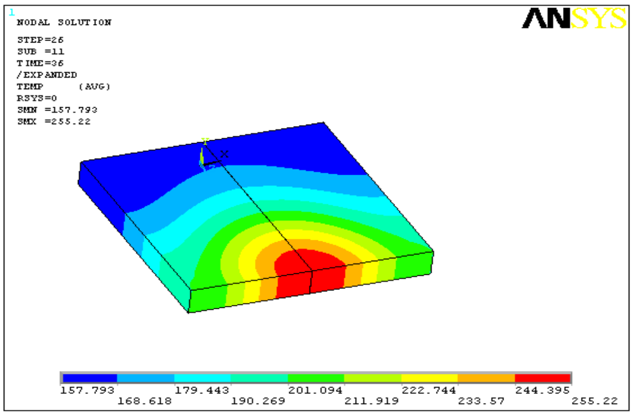

Fig4.4 Temperature distribution 10 seconds after completion of weld

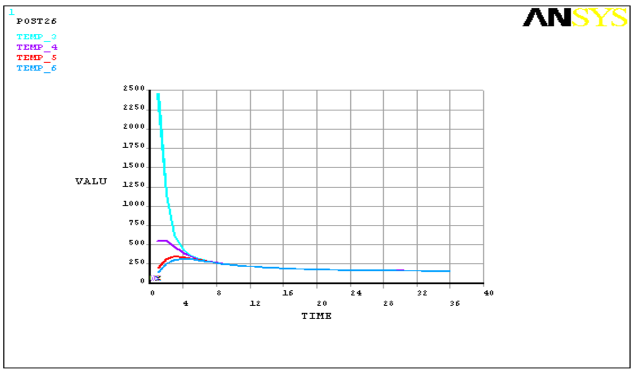

Fig 4.5 Temperature distribution along thickness at the start of weld process

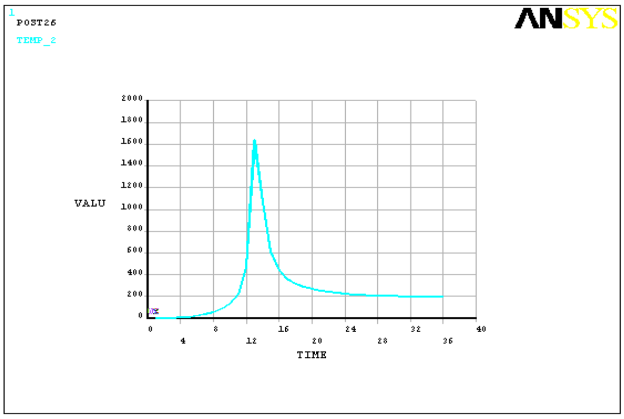

Fig 4.6 Temperature variation at the middle

Case 2: Heat Input : 5e7 w/m^2

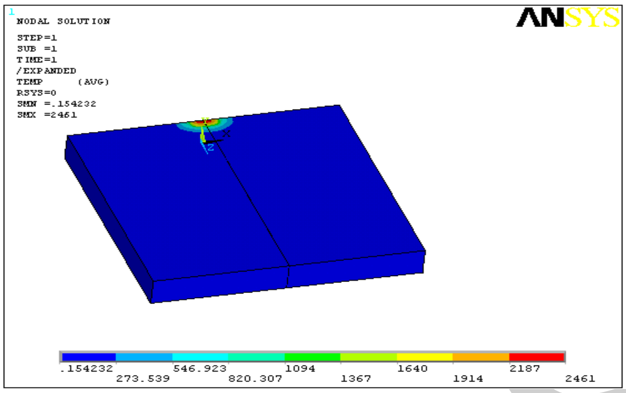

Fig 4.7 Temperature distribution at start of weld

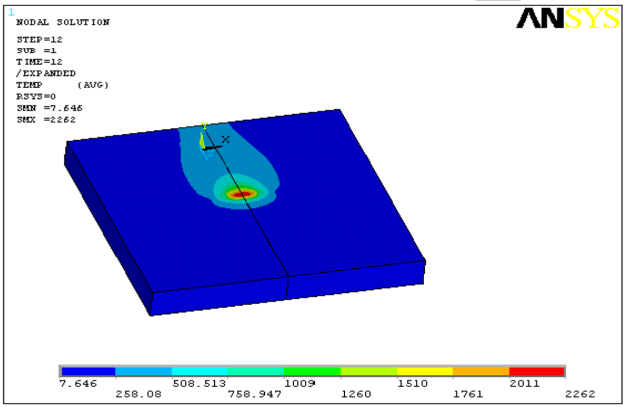

Fig 4.8 Temperature distribution after 12 sec of start of weld

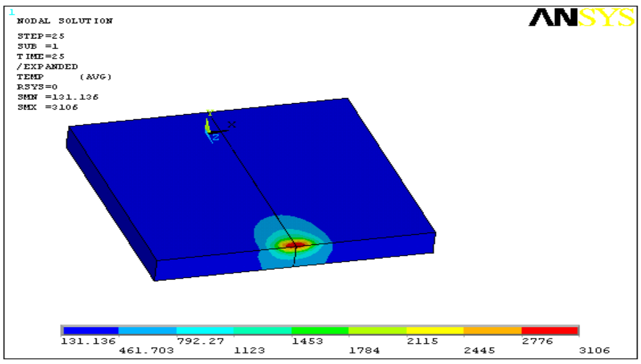

Fig 4.9 Temperature distribution after 25 sec of start of weld

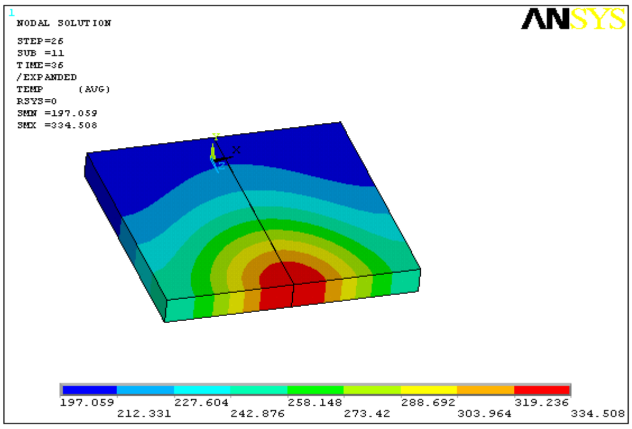

Fig 4.10 Temperature distribution after 10 secs

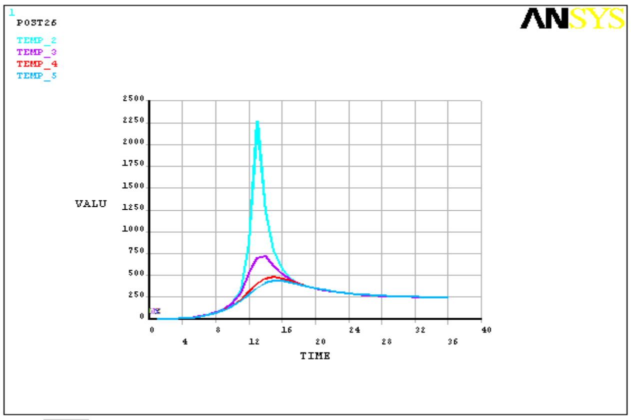

Fig 4.11 : Temperature variation at the middle

The figure 4.11 represents temperature variation at the middle. The graph represent higher temperature at the top surface compared to the bottom surface and temperature is dropping with increase in time.

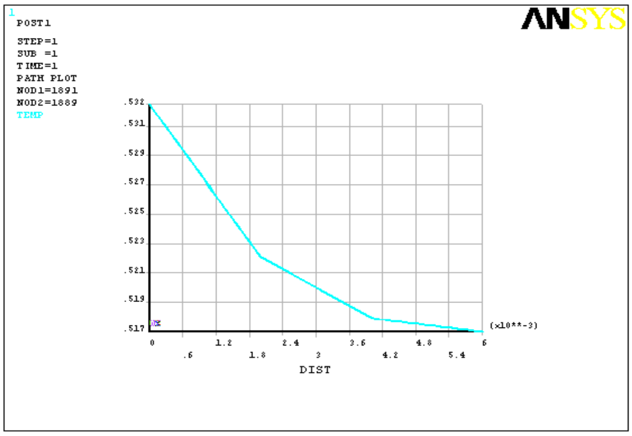

Fig 4.12 Temperature at the middle region

The Fig 4.12 shows dropping temperature across the thickness of the geometry. The graph values are taken at the center of the geometry. The top surface is maintained at 532 ^{0} c and the bottom is at 517 ^{0}c.

Structural results for different heat inputs

Case 1 : Heat Input 4e7 w/m2

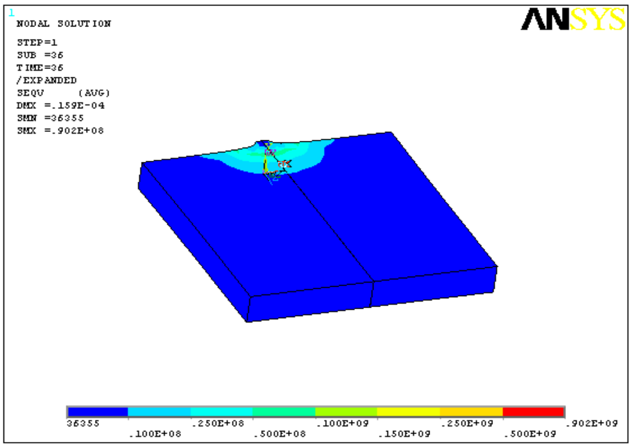

Fig 4.13 Vonmises stress in the structure

The above picture represents vonmises stress in the structure. Maximum vonmises stress can be observed at the starting end compared to other regions.

Case 2: Heat Input 5e7 w/m2

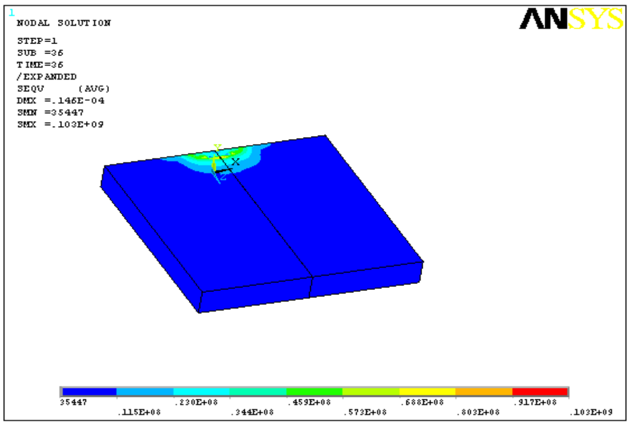

Fig 4.14 Vonmises stress in the structure

The Fig 4.14 represents vonmises stress in the structure. Maximum vonmises stress can be observed at the starting end compared to other regions.

V. CONCLUSION

The weld joint analysis is simulated using finite element methods and results are summarized as follows. Initially the weld plates are created taking the advantage of half symmetry using ansys top down approach.The structure is divided into 25 smaller parts for easier application of heat flux which is varying with time, The structure is meshed with 8 noded Solid70, a thermal element and convection loads are applied except at bottom and symmetrical surface and loading or welding zone. The problem is solved in transient domain for total 3t secs weld cycle time. The results are captured for thermal temperatures and residual stresses. Graphical plots are taken at the middle section to show variation of temperature and stresses. The graphical results shows higher temperature and stress value at the top surface compared to the bottom surface. But as the time increases, the difference of temperature is reducing between top and bottom surfaces. The problem has been analysed for 4 thermal heat flux values. The results shows increase of stress from 46 Mpa to 51 Mpa with increase in heat flux values from 4e7w/m^2 to 5.5w/m^2. So the results indicates higher heat flux increases the thermal streses.

REFERENCES

[1] MEDVEDEV,S.V.: Computer modeling of residual welding strains in technological design of welded structures , Welding International.-2002, Vol.16(1), pp.59-65.

[2] BONIFAZ,E.A.: Finite element analysis of heat flow in single-pass arc welds , Welding Res. Supplement, 2000, pp:121-125.

[3] DENG,D., MURAKAWA,H.: Numerical simulation of temperature field and residual stresses in multi-pass welds in stainless steel pipe and comparison with experimental measurements , Comput. Mater. Sci., 2006, 37:269-277.

[4] HANSEN,J.L.: Numerical modeling of we lding induced stresses , PhD thesis, Technical University of Denmark, ISBN 87-90855-52-3, 2003.

[5] JONSSON,M., KARLSSON,L., LINDGREN,L.E.: Deformations and stresses in butt-welding of large plates with special reference to the mechanical material properties , J. of Eng. Mat. and tech., 1985, Vol.107, pp.256-270.

[6] WIKANDER,L., KARLSSON,L., NASTROM,M., WEBSTER,P.: Finite element simulation and me asurement of welding residual Stresses , Modelling Simul.mater.Sci.Eng.,1994, Vol.2, pp.845-864.