Finite Element Analysis of Residual Stresses on Ferritic Stainless Steel using Shield Metal Arc Welding

Swapnil R. Deogade[1*], Prof. Sachin P. Ambade[2], Dr. Awanikumar Patil[3]

[1*]M.Tech Student (CAD/CAM), Department of Mechanical Engineering, Yeshwantrao Chavan College of Engineering, Nagpur
[2]Professor, Department of Mechanical Engineering, Yeshwantrao Chavan College of Engineering, Nagpur
[3]Professor, Department of Material and Metallurgical Engineering, Vishveshvarya National Institute of Technology, Nagpur

Corresponding author*
E-mail address: swapnildeogade@hotmail.com

Abstract- This work deals with the finite element analysis of Ferritic Stainless Steel using single pass Shielded Metal Arc Welding (SMAW). The analysis of stresses in heat affected zone (HAZ) and welded zone is carried out. The model uses FEA (ANSYS 14). Main purpose of this paper is to present both the methodologies of simulation by the ANSYS using coding carried out by mechanical APDL (ANSYS Parametric Design Language). The transient thermo-mechanical (coupled) analysis shows temperature and residual stress distributions on welded plates.

Keywords- Finite element analysis, Temperature distribution, Residual stress, ANSYS, SMAW

INTRODUCTION

Welding is most popular joining process utilized in various industries such as petrochemical, aero-space, automotive, marine etc. Welding is a versatile and relative low-cost process. However welding causes a non-uniform temperature distribution and produces residual stresses. This paper pertains to the evaluation of residual stress in 409M stainless steel (FSS) welding.

In the present analysis, the temperature distribution and the weld-induced residual stress fields and deformation of steel plates are investigated by numerical simulations based on FEM modeling using ANSYS14. The temperature dependent thermo-physical properties such as thermal conductivity, specific heat and density are provided and temperature dependent thermal–structural properties including Young's modulus, Poisson's ratio and thermal expansion coefficient are used for thermal analysis and mechanical analysis, respectively. The heat flux provided in thermal analysis was calculated using Goldak's heat source model. Finite element analysis to simulate the transient thermal conditions of the weld is considered to be the most accurate and flexible method of modeling.

LITERATURE REVIEW

S. Murugan et all [1], studied the Temperature distribution and residual stresses due to multipass welding in type 304 stainless steel and low carbon steel weld pads. In a multipass welding operation, the residual stresses are developed. This change stresses with every weld pass. Among various welding operation they carried out MMAW i.e., Manual Metal Arc Welding. This tensile residual stresses increases susceptibility of weld to fatigue damage, stress corrosion cracking as well as fracture. M. Jeyakumar et al [2] did the evaluation of residual stress in butt-welded steel plates. The residual stresses and distortions are dominated by deformation of metals in the heat affected zone of weld joints as well as by external and internal restraints. The residual stress effects may be either beneficial or detrimental which depends upon magnitude and distribution of stresses. Since the load steps are more mechanical APDL is adopted over here. The analysis results found in this research is in good agreement with existing complex 3D finite element analysis and experiments. K Punitharani, et al [3] discussed the Finite element method for residual stresses and distortion in hard faced gate valve. The process of depositing a filler material on the surface of carbon and low alloy steel base metal is called hard facing. In this work residual stresses are predicted in hard face gate valve using FEA and with the help of X-ray diffraction technique stresses measured are being validated. Here the load steps fairly are very high, therefore programming language called ANSYS parametric design language is used and the coding was employed to perform both thermal and structural analysis.

C.M. Chen and R. Kovacevic [4] explained the finite element modeling of friction stir welding thermal and thermo-mechanical analysis. Friction stir welding is nowadays is emerged as solid state joining. Here it is expected that the residual stresses and distortions generated are less as compared with fusion welding. G.A.Moraitis and G.N.Labeas [5] made the prediction of residual stresses and distortions due to laser beam welding of butt joints in pressure vessels. A 3 dimensional model has been developed for the simulation of laser welding process and later predicting the distortions and residual stresses of laser beam welding steel and aluminum butt joints. Simulation model helps us to control residual stresses and distortion within the welded structures. Ali Moarrefzadeh [6] investigated Finite Element simulation for thermal profile in shielded metal arc welding (SMAW) process. FEM simulation can provide detail information of stress distribution, deformations and temperatures.In finite element simulations two numerical formations were there viz., Lagrangian formulation and Eulerian formulation. For meshing of solid field PLANE55 type is used which has 4 nodes with one degree of freedom. This element has mesh moving property as well. For fluid field FLUID141 element is used.

J.J. del et all [7] made the Comparative analysis of TIG welding distortions between austenitic and duplex stainless steels by FEM. The aim of this paper was to establish a set of approximations and simplification in numerical modelling of stainless steel welding for reducing huge computation time due to coupled and nonlinear formulations. N. Akkus, G. Genc, and S. Sen [8] showed experiments and Finite Element Analysis of Arc Welding Residual Stresses. In sheet metal after welding residual stresses occurs so in this paper FEA and experimentation is carried out. Thermocouples are used for temperature measurement. Residual stresses are investigated with the help of hole drilling method. In residual stress measurement experiment as the welding speed increases less strain and less residual stresses were observed. Li Chaowen and Wang Yong [9] explained three-dimensional FEA of temperature and stress distributions for in-service welding process. Welding onto a pipeline in active operation, called in-service welding. It is the advanced technique utilized in repair of pipelines. These in-service welding repair methods are widely used through the natural gas, petroleum and petrochemical industries. Shielded metal arc welding was carried out here as a heat source model and the doubled ellipsoid model was selected. For verifying the FE model, the temperature obtained by the thermocouples was compared to the EF calculated. Dragi Stamenkovic and Ivana Vasovic [10] studied about Finite Element Analysis of Residual Stresses in Butt Welding Two Similar Plates. The joining of dissimilar metals are more challenging than that in similar metals due to difference in properties of base metals welded. Therefore welded structures first meet the strength requirements and probability of defect formation. There are two different methods in couple-field analysis viz., sequential and direct. Fanrong Kong and Radovan Kovacevic [11] worked on 3D finite element modelling of the thermally induced residual stresses in the hybrid laser/arc welding of lap joint. In this study, four cases of hybrid laser–GTAW experiment with different welding speeds of 20 mm/s, 25 mm/s, 30 mm/s, and 40mm/s are performed to validate the thermo-mechanical finite element model. The finer mesh is placed near and along the weld bead in order to assure enough accuracy in simulation, and the courser mesh is chosen for the areas far from the weld bead in order to reduce the computation cost. All the procedure was carried out using ANSYS program designed language (APDL).

FINITE ELEMENT ANALYSIS

The finite element method is a numerical procedure that can be used to obtain solutions to a large class of engineering problems involving stress analysis, heat transfer, electromagnetism, and fluid flow. ANSYS is a comprehensive general-purpose finite element computer program that contains over 100,000 lines of code. ANSYS is capable of performing static, dynamic, heat transfer, fluid flow, and electromagnetism analyses. In order to accurately capture the temperature fields and the residual stresses in the welded pipe, a 3-D finite element model is developed. The thermo-mechanical behavior of the weldment during welding is simulated using coupled formulation.

Thermal Analysis

Heat conduction is assumed governed by the Fourier law. Together with the source term from the process, the (transient) governing equation for temperatures becomes

$$\rho c_p \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) + \dot{Q}_v \qquad 1$$

To analyse the transient energy distribution in a material, conductivity and heat capacity of the material must be specified.

Material Model

In the present thermal analysis, two work-pieces of dimension 150*75*4 were developed in APDL and were glued. The material is meshed using a brick element called SOLID70 which is 8 noded three dimensional element. Meshing was carried out was mapped meshing which was fine at welded zone and coarse at heat affected zone. The element is defined by eight nodes with temperature as single degree of freedom at each node and by the orthotropic material properties. The Goldak moving heat source was modeled in ansys.

Boundary conditions for thermal model were specified as surface loads through ANSYS® codes. Assumptions were made for various boundary conditions based on data collected from various published research papers. Convective and radiative heat losses to the ambient occurs across all free surfaces of the workpiece and conduction losses occur from the workpiece bottom surface to the backing plate. According to Stefan-Boltzmann's law the radiative heat loss is given by

$$q_{rad} = \varepsilon\sigma((T - T_z)^4 - (T_0 - T_z)^4) \qquad 2$$

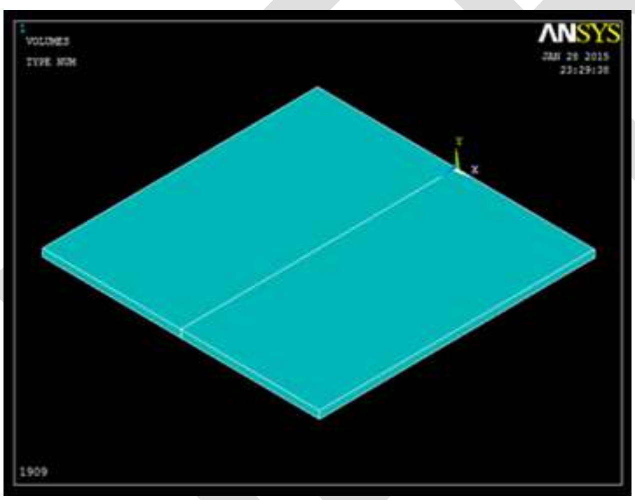

Fig. 1. Model developed in APDL

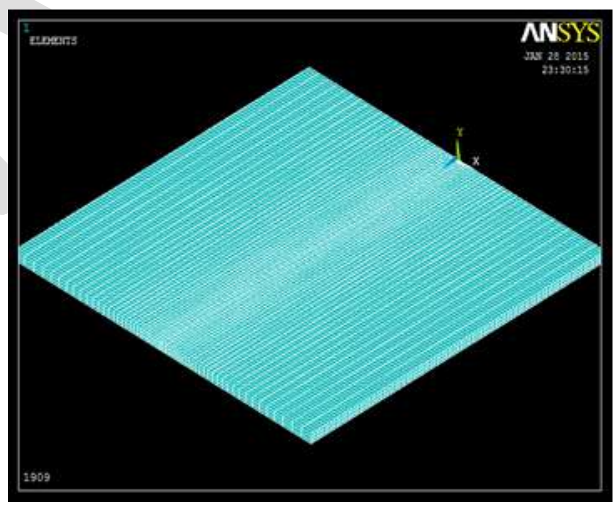

Fig. 2. Meshed model

Structural Analysis

The same finite element model used in the thermal analysis was employed here, except for the element type and the boundary conditions. The mechanical analysis is conducted using the temperature histories computed by the thermal analysis as the input data. During the welding process, solid-state phase transformation does not occur in the stainless base metal and the weld metal. The elastic strain is modeled using the isotropic Hook's law with temperature-dependent Young's modulus and Poisson's ratio.

ISSN 2091-2730

Here the coupled thermo-mechanical analysis was carried out. The results from thermal analysis were added as the input to structural and hence no boundary conditions were provided to it. Only the material properties like Modulus of elasticity, poisons ratio and thermal expansion coefficient were provided to it. The element Solid 70 is replaced automatically by the equivalent structural element Solid 185, which is also an eight-noded, three-dimensional element but has plasticity, hyper-elasticity, stress stiffening, creep, large deflection, and large strain capabilities.

RESULTS AND DISCUSSIONS

Thermal profile of welding using ANSYS

Using thermal analysis movement of heat source at various times and temperature distribution are shown below in figure 3 and 4. Heat source starts at 0 and continues upto to 66 seconds, since the welding time from start to end was 66 seconds. Below are the heat sources shown in figure 3 are after (a) 11seconds, (b) 22seconds, (c) 44seconds and (d) 65.56seconds. With the help of Goldak's heat source model heat flux are estimated and were applied at the welded zone. (were fine mesh were provided.)

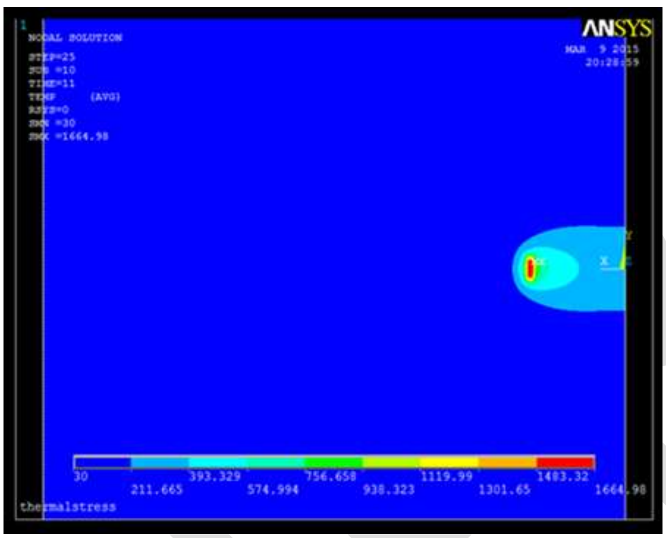

Fig. 3(a). At 11 seconds

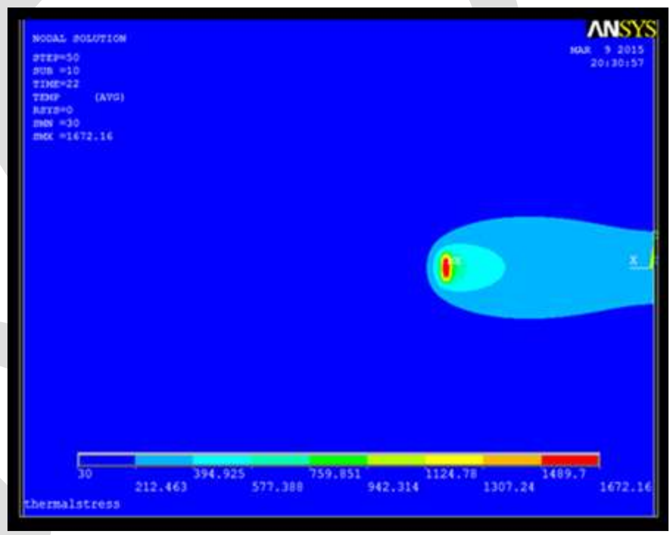

Fig. 3(b). At 22 seconds

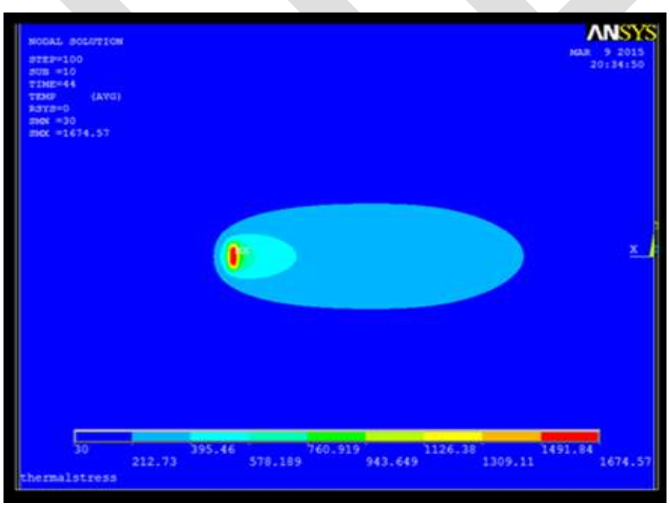

Fig. 3(c). At 44 seconds

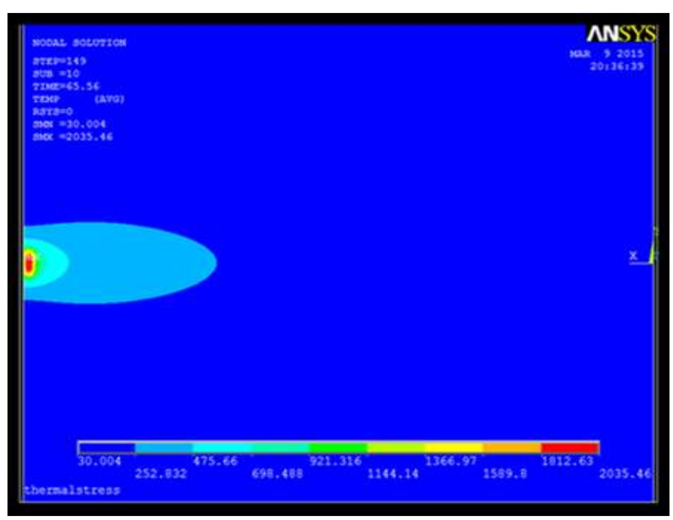

Fig. 3(d). At 65.56 seconds

Fig. 3. Movement of heat source at various times

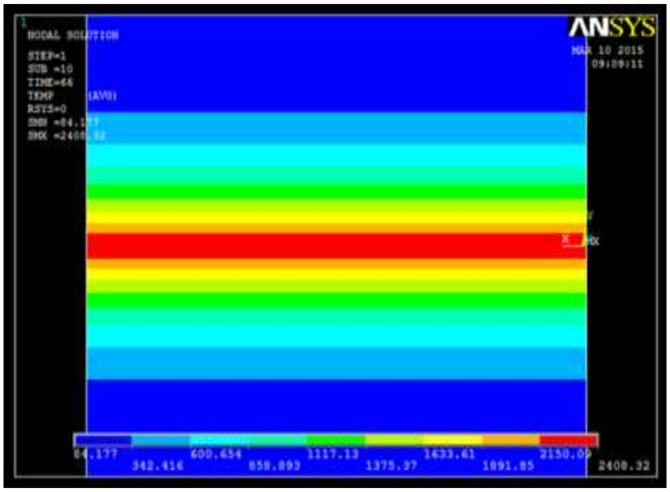

Fig. 4. Temperature distribution

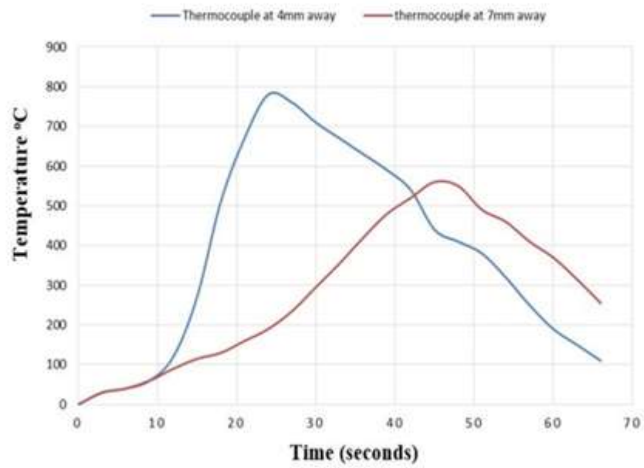

Fig. 5. Thermal profile using ANSYS

Two thermocouples were inserted in plates at mm and 7mm away from welding inorder to find out the temperature profile at heat affected zone from starting point of welding. The thermocouples were inserted at a distance of 50mm and 100mm. Following figure 5. shows the temperature distribution at heat affected zone as the welding torch moves.

Structural Analysis

As the structural analysis is coupled with thermal analysis, hence carried out after the thermal analysis. It does not require boundary conditions. Outputs from thermal results are treated as input to structural analysis. As no boundary conditions are provided therefore displacement is considered to be zero. Below shown are the stresses which are developed in the work-piece after welding. Figures 6(a,b,c) shows the stresses in x, y and z directions, Figures 7(a,b,c) shows the shear stresses in x, y and z directions. Figure 8 shows the equivalent stress.

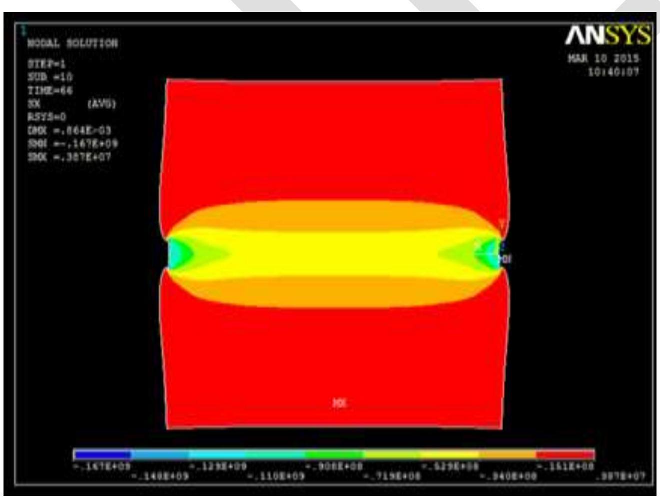

Fig. 6(a). Stress in x-direction

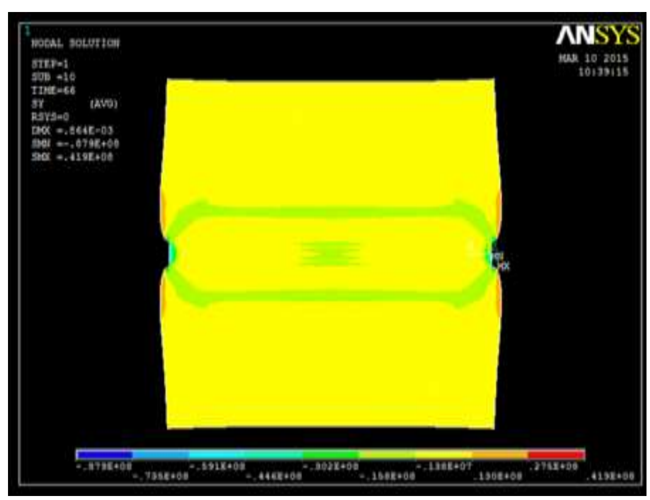

Fig. 6(b). Stress in y-direction

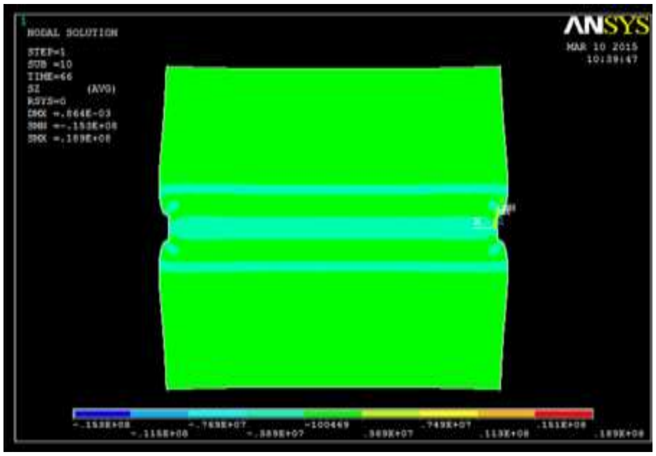

Fig. 6(c). Stress in z-direction

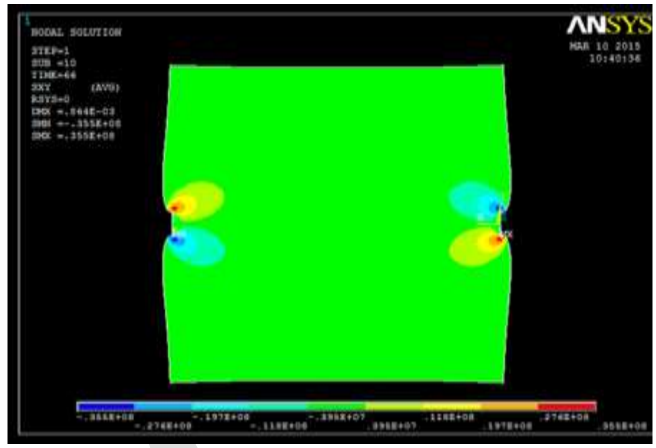

Fig. 7(a). Shear stress in x-direction

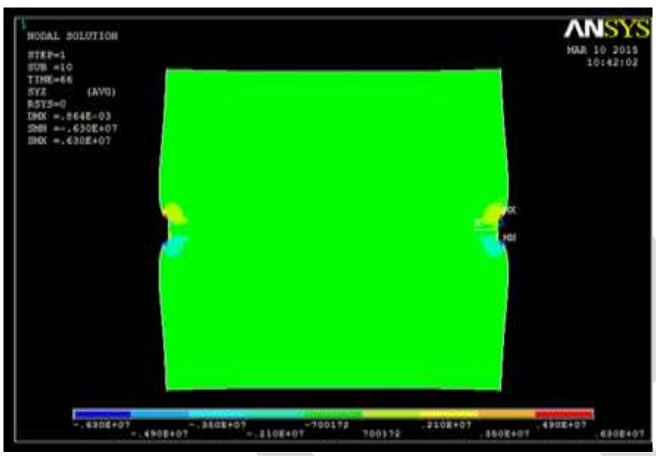

Fig. 7(b). Shear stress in y-direction

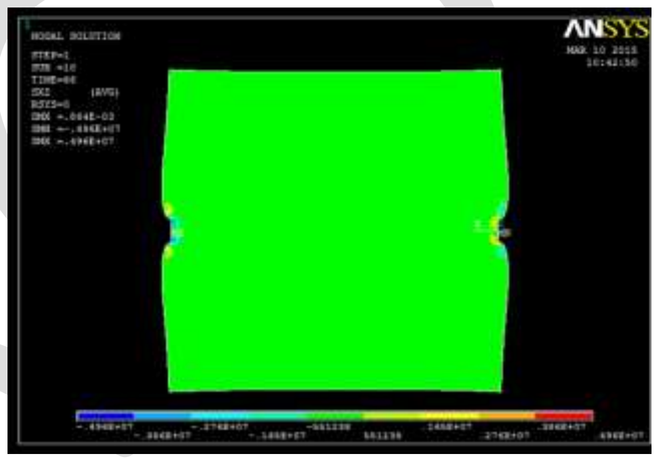

Fig. 7(c). Shear stress in z-direction

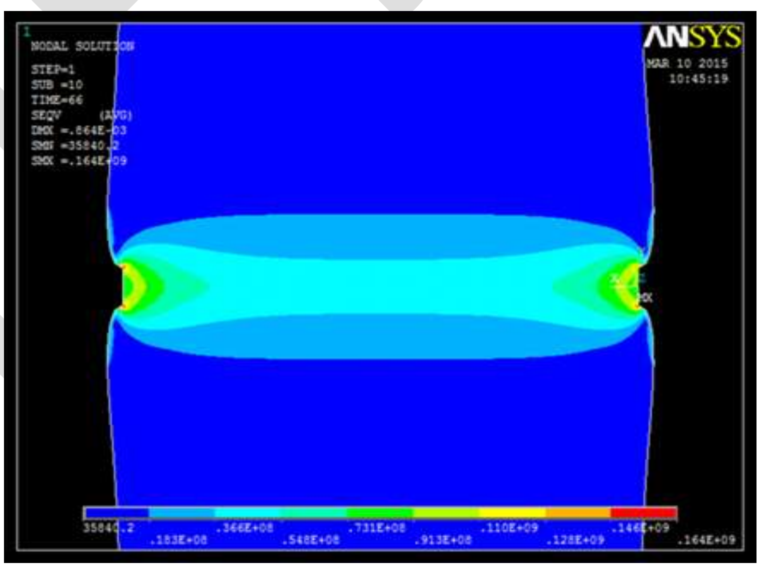

Fig. 8. Von-mises stress

CONCLUSION

Finite Element Analysis of stresses has been carried out ANSYS 14. In this study 3-D FE model is developed to analyze the temperature fields and stress distribution for FSS409M. The 3-D Finite Eelement model which was developed have predicted temperature cycles and welding residual stress fields satisfactorily.

REFERENCES:

1. S. Murugan, Sanjai K. Rai, P.V. Kumar, T. Jayakumar, Baldev Raj, M.S.C. Bose; "Temperature distribution and residual stresses due to multipass welding in type 304 stainless steel and low carbon steel weld pads", International Journal of Pressure Vessels and Piping 78 (2001) 307-317
2. M. Jeyakumar, T. Christopher, R. Narayanan and B. Nageswara Rao; "Residual Stress Evaluation in Butt-welded IN718 Plates", CJBAS Vol. (01) - September - Issue 02 (2013) 88-99
3. K Punitharani, N Murugan and S M Sivagami; "Finite element analysis of residual stresses and distortion in hard faced gate valve", Journal of scientific and industrial research, vol. 69 February 2010, pp. 129-134
4. C.M. Chen, R. Kovacevic; "Finite element modeling of friction stir welding-thermal and thermo-mechanical analysis", International Journal of Machine Tools & Manufacture 43 (2003) 1319–1326
5. G.A. Moraitis, G.N. Labeas; "Residual stress and distortion calculation of laser beam welding for aluminum lap joints", Journal of materials processing technology 198 (2008) 260–269
6. Ali Moarrefzadeh; "Finite-Element Simulation for Thermal Profile in Shielded Metal Arc Welding (SMAW)Process", International Journal of Emerging trends in Engineering and Development ISSN 2249-6149, Issue 2, Vol.1(January-2012)
7. J.J. del Coz Díaz, P. Menéndez Rodríguez, P.J. García Nieto, D. Castro-Fresno; "Comparative analysis of TIG welding distortions between austenitic and duplex stainless steels by FEM", Applied Thermal Engineering 30 (2010) 2448-2459
8. N. Akkuş, G. Genç, S. Şen; "Experiments and Finite Element Analysis of Arc Welding Residual Stresses", International Congress on Advances in Welding Science and Technology for Construction, Energy and Transportation Systems (AWST - 2011) 24-25 October 2011, Antalya, Turkey
9. Li Chaowen and Wang Yong; "Three-dimensional finite element analysis of temperature and stress distributions for in-service welding process", Materials and Design 52 (2013) 1052–1057
10. Dragi Stamenković, MSc (Eng)1),Ivana Vasović, BSc (Eng); "Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates", Scientific Technical Review, Vol.LIX,No.1,2009, pp. 57-60
11. Fanrong Kong, Radovan Kovacevic; "3D finite element modeling of the thermally induced residual stress in the hybrid laser/arc welding of lap joint", Journal of Materials Processing Technology 210 (2010) 941–950