# THERMO-MECHANICAL ANALYSIS OF A CORNER WELDED JOINT BY FINITE ELEMENT METHOD

[1] K.Ashok Kumar [2] Dr. CH.Srinivasa Rao [3] T.N Charyulu [4] SCV Ramana Murthy Naidu

[1] M.Tech. Scholar, Nova College of Engineering &Technology, Jangareddygudem, West Godavari. A.P.

[2, 4] Department of Mechanical Engineering, KHIT,Guntur,A.P.

[3] Head, Department of Mechanical Engineering, Nova College of Engineering &Technology, Jangareddygudem, West Godavari, A.P.

## ABSTRACT

Arc welding is one of the mostly used fabrication process in the industry due to its potent applications. During manual metal Arc Welding heat is transferred to the joint by melting an electrode. During welding process, the thermal stresses are generated in the specimen causing distortion and change in the shape. This problem of residual stresses and distortion of welded plates has become a major issue of research on shipbuilding and machine tool bodies.

In the present work a corner welded joint has been used to simulate the Arc Welding process for the purpose of estimation of distortion and residual stresses. The simulation has been carried using a numerical technique Finite Element Method and with the commercial software ANSYS 11.0. The FEM has been carried in two stages. In the first stage, the pure thermal analysis is carried to estimate the time dependent temperatures. The temperatures obtained in thermal analysis are given as input in the second stage i.e. The Structural Analysis. The output of this thermo mechanical analysis involves estimation of Von mises Stresses, Strains and deformation.

**Key words:** Arc Welding, Finite Element Method, Residual Stresses & Deformation.

## INTRODUCTION AND BACK ROUND

In welding, two or more pieces of metal are joined together by the application of heat, pressure, or a combination of both. Most of the processes are grouped into two main categories: pressure welding, in which the weld is achieved by pressure; and heat welding, in which the weld is achieved by heat. Heat welding is the most common welding used today.

Arc welding, which is heat-type welding, is one of the most important manufacturing operations for the joining of structural elements for a wide range of applications e.g. guide way for trains, ships, bridges, building structures, automobiles, and nuclear reactors etc. In Arc welding, either direct or alternating electric current is continuously supplied to create an electric arc that generates enough heat to melt the metal and form a weld.

The research activity in welding simulation started decades ago. An analytical solution of heat flow during welding process has been developed based on conduction heat transfer for predicting the shape of the weld pool [1]. The FEM is a proven technique to simulate the complex welding process [2]. Over the past few years, Finite Element Method (FEM) has been used extensively to predict distortion and residual stresses due to welding operations [3,4]. The work done by the Friedman et al has been verified by 3-D computational modeling and the results are in good agreement [5].

The transient temperature distributions and temperature variations of the welded plates during welding were predicted and the fusion zone and heat affected zone were obtained [6]. A space–time FEM is proposed to solve the transient convection–diffusion thermal equation [7]. The validity of the numerical model is confirmed by comparing the simulation results with the corresponding experimental findings [8]. A three-dimensional transient model is developed to analyze the periodic changes of the temperature field, weld pool and keyhole shape and dimensions during the controlled pulse key-holing process [9].

The problem of welding distortion during large steel fabrications leads to dimensional inaccuracies and misalignments of structural members, which can result in corrective tasks or rework when tolerance limits are exceeded. This in turn, increases the cost of production and leads to delays.

Therefore, the problems of distortion and residual stresses are always of great concern in welding industry. Even though ample of work has been carried on welding phenomenon still there is a lacuna in the study of the effects of heat input, welding speed, restraint, plate curvature, and gap on arc welding responses. Thus the objective of the present paper is to simulate the complex arc welding process using the finite element code ANSYS and study the Von mises Stresses, Strains, and distortion due to thermo-mechanical response.

## FINITE ELEMENT MODELING:

The FE modeling consists of geometry construction, meshing, loading and boundary condition, simulation and port processing results. Two number 3-dimensional rectangular plates are used to obtain an L-shape corner joint. The dimensions of both the horizontal and vertical plates are of length of 0.28m, width 0.1m and thickness of 0.006m. The temperature-dependent Material Properties as shown in Table –I are given as input.

The solid 70 and solid 185 elements have been used for meshing. The solid 70 is used in the thermal analysis and solid185 used in Structural Analysis. The number of elements is 1391. The number of element divisions in X&Y Directions are 10 and along Z-Direction is 28. The model with the FE mesh has been shown in Figure-1.

**Table-1: Temperature-Dependent Material Properties**

| Temperatures (Celsius) | Modulus of Elasticity (MPa) | Poisson Ratio | Thermal Ex. Coef. ( $10^{-6}$ /° C) | Thermal Conductivity (W/m° K) | Specific Heat (J/kg° K) |
|---|---|---|---|---|---|
| 0 | 314 | 0.2786 | 10 | 51.9 | 450 |
| 100 | 349 | 0.3095 | 11 | 51.1 | 499.2 |
| 300 | 440 | 0.331 | 12 | 46.1 | 565.5 |
| 450 | 460 | 0.338 | 13 | 41.05 | 630.5 |
| 550 | 410 | 0.3575 | 14 | 37.5 | 705.5 |
| 600 | 330 | 0.3778 | 14 | 35.6 | 773.3 |
| 720 | 58.8 | 0.3738 | 14 | 30.64 | 1080.4 |
| 800 | 58.8 | 0.4238 | 14 | 26 | 931 |
| 1450 | 1.29 | 0.4738 | 15 | 29.45 | 437.93 |
| 1510 | 1.0 | 0.499 | 15 | 29.7 | 400 |
| 1580 | 0.01 | 0.499 | 15 | 29.7 | 735.25 |
| 5000 | 0.01 | 0.499 | 15.5 | 42.2 | 400 |

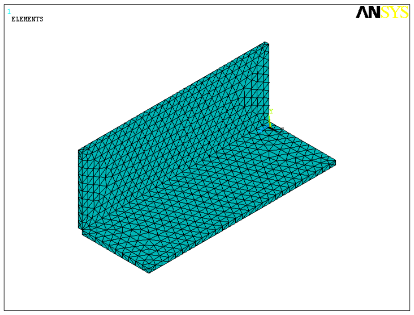


Figure-1: FE Mesh of the model

A heat input of 1200 W is given at the welded joint i.e. between the plates along the centre line. Convection is applied on the exposed surface with a film coefficient of h=5000 w/$m^2$ $^0$C and the bulk temperature of 25$^0$C. The surface temperature of the specimen is Zero. The simulation has been carried for thermal analysis in a time period of 1200 seconds. The number of sub steps are 20 the time at the end of load step is 1200seconds on the size of the time step is 60. The Newmark algorithm has been used to run the simulation with transient effects. The temperatures obtained from thermal analysis are imported in to structural analysis for getting thermal stresses and strains.

## RESULTS& DISCUSSION

The results and discussions drawn from the present work include the temperature distribution with respect to time in thermal analysis and estimation of Von mises stresses, Strains, Elastic Strains, Total mechanical strains by combined Thermo-mechanical analysis.

### Thermal Analysis:

In transient thermal analysis, the temperature distribution in the specimen is estimated. The temperature variation in all(X, Y &Z) directions with respect to time is as shown in Figure 2.

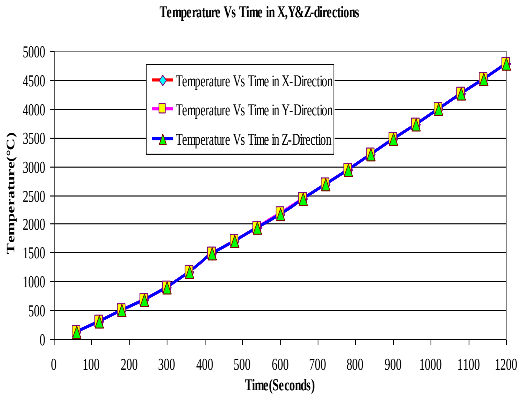


**Figure-2: Temperature Vs Time**

From the Figure, it can be observed that the temperature is varying uniformly and is rising with respect to time in all the directions. The temperature rises initially at a slower rate and rapid rate later but the rate is uniformly increasing. It is because initially the work piece is at cold condition and takes more time to transfer the heat. During continuation of welding due to less potential difference the temperature raises at a faster rate. The temperature results obtained by thermal analysis are given as input to the structural analysis for performing thermo-mechanical analysis.

### Thermo-mechanical Analysis:

The results of thermo mechanical analysis include Von mises stresses (Thermal Stresses), thermal strains, deformation or displacements, Elastic Strains and total mechanical strains. The results also include the deformed and undeformed contours due to thermal stresses with data tables.

**Displacements:** The deformation of the corner joint due to thermal stresses generated is given by the displacements X, Y&Z directions. The displacement of the horizontal plate in X, Y&Z directions with respect to the dimensionless parameter(X/L) is shown in Figure-3. The dimensionless parameter is an account of distance from the origin at any point X in the length of the specimen as shown in Figure-3.

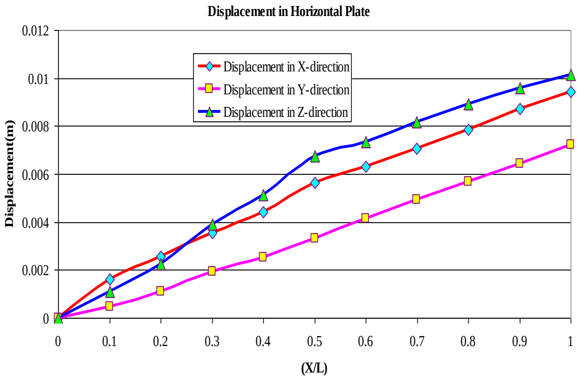


**Figure-3 Displacement in Horizontal Plate**

The deformation is more in Z-direction compared to X& Y directions for the horizontal plate. This is due to the fact that the longitudinal strain is more than lateral strain. As shown in Figure-3 the displacement in Z-direction is continuously increasing from line of welding to the other end. Almost the same trend is obtained in X&Y directions. This is occurring because of heat transfer process normal to the direction of thickness. The highest deformation is found to be 0.01m at the extreme opposite end of the origin in Z-direction. The deformation of the horizontal plate is as shown in Figure-4.

**Figure-4 Deformation of the horizontal plate**

Similarly the deformation of the vertical plate of the corner joint in X, Y& Z directions is shown in Figure-5. The highest deformation is found to be 0.01m at the extreme opposite end of the origin in Z-direction.It is observed that the displacement in horizontal and vertical plates is following same trend due to of similarity on either side of weld line. The deformation of the Vertical plate is as shown in Figure-6.

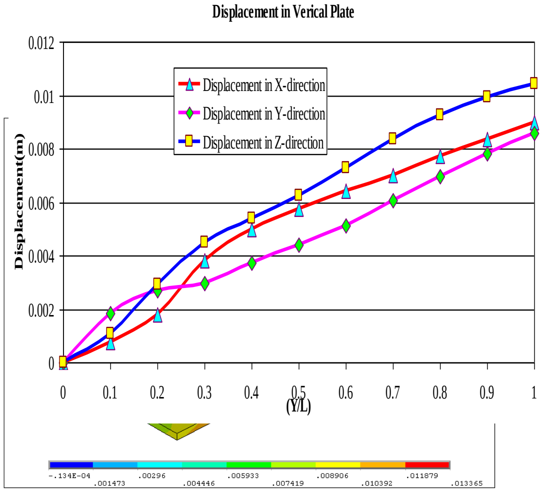


**Figure-5: Displacement in Vertical Plate**

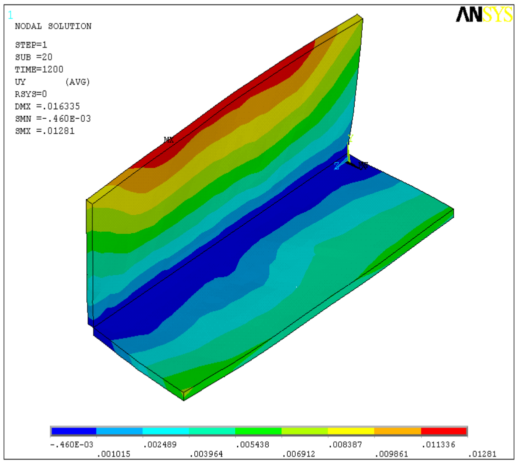


**Figure-6: Deformation of the Vertical plate**

**Von mises Stresses:** The variation of Von mises Stresses in X, Y & Z directions in the horizontal plate is as shown in Figure7. The maximum stresses are found in X-direction of the horizontal plate in the corner joint.

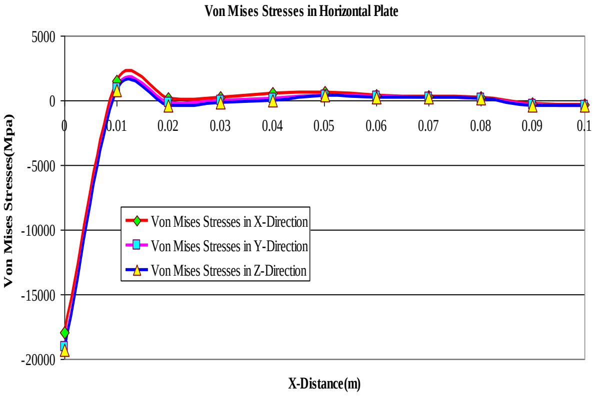
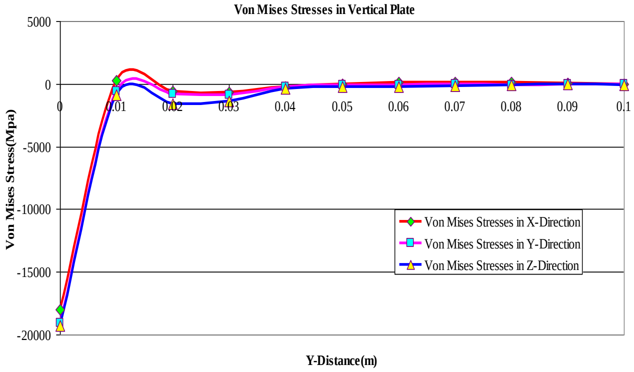


**Figure-7 Von mises Stresses in horizontal Plate**

The highest stress value of 1510 Mpa is found at a distance of 10mm from the origin in X-direction and a minimum value of -315 Mpa compressive stresses at the extreme end of the horizontal plate. The highest compressive stresses are produced at the origin as indicated in the Figure. This is due to the accumulated effect of temperature raise at the welded joint and more heat is generated at the origin where the heat flux (by melting the electrode) is given as heat input. This is also due to neglected exposed surfaces temperature boundary conditions and heat loss only by convection. Comparing with stresses in X-direction, the Y&Z direction stress are relatively less and is due to heat input given in horizontal direction for the horizontal plate.



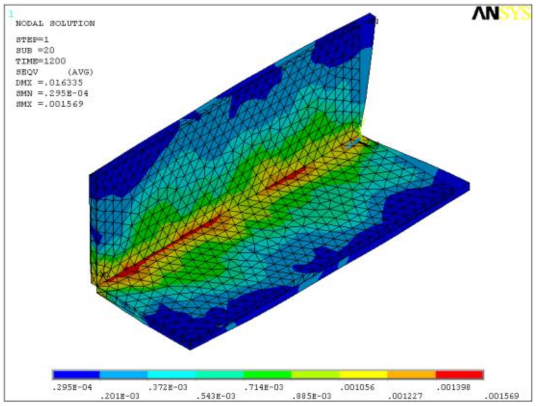


Figure-8: Von mises Stresses (X, Y &Z directions)

**Von mises Stresses in Vertical Plate:**

The variation of Von mises Stresses in X,Y & Z directions in the vertical plate is as shown in Figure9. The maximum stresses are found in X-direction of the vertical plate in the corner joint. The highest stress value of 249 Mpa is found at a distance of 10mm from the origin in X-direction and a minimum value of -1800 Mpa compressive stresses at the extreme end of the vertical plate. Highest compressive stresses are produced at the origin as indicated in the figure. The cumulative effect of temperature raise at the welded joint heat flux is given as, at the origin is responsible for it. Comparing with stresses in X-direction the Y&Z direction stress are relatively less due to heat input is given vertical direction for the vertical plate.

**Figure-9: Von mises Stresses in Vertical Plate**

### Von mises Stresses along Z-direction:

The variation of Von mises Stresses in X, Y & Z directions along length is as shown in Figure10. The maximum thermal stresses are estimated at the end of the chosen time period of 1200 seconds.

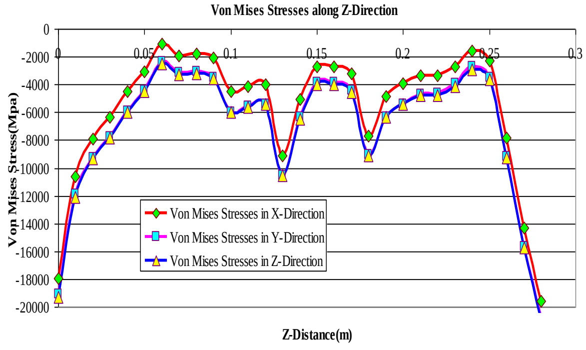


**Figure10: Von mises Stresses along Z-direction**

The highest value of thermal stress is -1060 Mpa with restricted displacement along the joint.

## Conclusion:

The thermo-mechanical response of the complex Arc welding of a corner joint has been successfully simulated by Finite Element Modeling. The effect of heat input direction and time dependent material properties on distortion of welded joint has been studied. The temperatures obtained by thermal analysis have been used to obtain stress-strain values and distortion during combined thermo-mechanical analysis. The obtained thermal stresses are is good agreement with theoretical values and the results have been discussed by considering the phenomenal changes during welding.

## References:

[1]**.** Rosenthal D. “The Theory of moving Sources of Heat and Its Application to Metal Treatments”, Transactions of A.S.M.E., Nov. 1946, p 849-866.

[2]. Friedman E., “Thermomechanical Analysis of the Welding Process Using the Finite Element Method”,

Transaction of the ASME, 1975, p 206-213.

[3]. Brown S., and Song H., "Finite Element Simulation of Welding of Large Structure", Journal of Engineering for Industry, vol. 144, 1992, p 441-451.

[4]. Michaleris P. and Debiccari A., "Prediction of Welding Distortion", Welding Journal, April 1997, p 172-180.

[5]. Taylor G., Hughes M., Strusevich N., and Pericleous K., "Finite Volume Methods Applied to the Computational Modeling of Welding Phenomena", Proceedings, Conference on CFD in the Minerals and Process Industries, Melbourne, Australia, Dec. 1999, pp.1-6.

[6]. D. Gery , H. Long and P.Maropoulos" Effects of welding speed, energy input and heat source distribution on temperature variations in butt joint welding" August 2005, pp.393–401.

[7]. Shaodong Wang and John Goldak "Simulation on the thermal cycle of a welding process by space–time convection–diffusion finite element analysis" May 2009, pp. 936–947.

[8]. Hwa Teng Lee, Chun Te Chen and Jia Lin Wu "Numerical and experimental investigation into effect of temperature field on sensitization of Alloy 690 butt welds fabricated by gas tungsten arcwelding and laser beam welding" September 2010, pp.1636–1645

[9]. Junhua Sun, Chuan Song Wu and Yanhui Feng "Modeling the transient heat transfer for the controlled pulse key-holing process in plasma arc welding" September 2011, pp.1664–1671.