RESEARCH ARTICLE OPEN ACCESS

Finite Element Simulation of Plasma Transferred ARC Welding [PTAW] of Structural Steel

PV Senthiil* , Aakash Shirrushti **
* Head, Mechanical Engineering, SPIHER, St.Peters University,Chennai-600054,
**Department of Mech, SRM University, Chennai

ABSTRACT
Plasma transferred Arc welding is one of the most widely used welding process, in which the metals are fused just above the melting point, and makes the metal to fuse. It is employed in many applications like tool die and metal casting, strip metal welding etc. This investigation is to analyze temperature distribution residual stress and distortion by varying the heat source parameter in SYSWELD, and compared the results with ANSYS. The simulation of Plasma Transferred Arc welding was of structural steel plate performed using a non-linear transient heat transfer analysis. Heat losses due to convection and variation of material properties with temperature were considered in this analysis. To incorporate the heat developed the Gaussian distribution was considered. Finite element simulations were performed using ANSYS Parametric Design Language (APDL) code and using SYSWELD. The temperatures obtained were compared with experimental results for validation. It was found that the predicted values of temperature agree very well with the experimental values. Residual Stress and Distortion were also predicted for various heat Input. The effect of heat input on residual stress and distortion was investigated.
Keywords – residual stress, heat transfer, plasma weld, distortion,PTAW

I. INTRODUCTION

The PTAW welding process and equipment are a modification of Gas Tungsten Arc Welding (GTAW). The significant difference involves a redesign of the GTAW torch to provide a concentration and collimation of the arc plasma. This is achieved by generating the arc and its plasma within the confines of the torch followed by discharging the high temperature ionized plasma through an orifice. The diameter, configuration and throat length of the orifice are carefully designed to maximize the desirable properties of the arc, i.e. culmination, high temperature and stability. Orifice details are interchangeable so that variations in these dimensions are available to satisfy a variety of weld requirements. The orifice concentrates the plasma arc and collimates the flow so that it assumes a beam configuration as opposed to the "umbrella" shaped arc of the GTAW process.[5]

As a result PTA welds have a deeper penetration with reduced heat-affected zones than can be achieved with GTAW. Both of these properties are attributed to the effect of energy concentration and therefore input rate and the reduction of peripheral heating affects (thermal radiation and convection). Because the arc is collimated arc length is also less critical than that of GTAW. This is a distinct advantage when the arc length varies due to weld joint surface and dimensional irregularities.

1.1 PRINCIPLE OF PTAW

Plasma welding with transferred arc mode is shown in Figure 1, employing a gas shielded arc between non-consumable tungsten electrode and base metal. If an electric arc between a tungsten electrode and the work piece is constricted in a cross-sectional area, its temperature increases and the arc is focused through a specially designed nozzle, called plasma. The orifice inert gas becomes ionized in the nozzle and provides a stable discharge path towards the work piece even at low power.

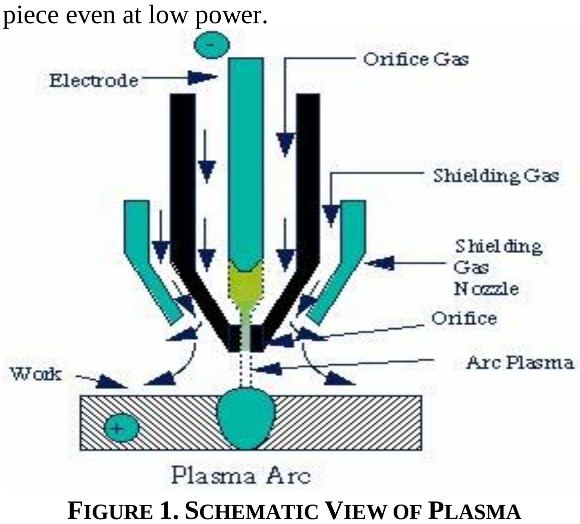


FIGURE 1. SCHEMATIC VIEW OF PLASMA TRANSFERRED ARC WELDING (WWW.JOININGTECH.COM)

1.2 OBJECTIVES OF THE PROJECT

The main objectives of this research are

1. To simulate the Plasma Transferred Arc Welding of Structural Steel using SYSWELD and ANSYS.
2. To vary the heat source parameter applied in SYSWELD and to find the temperature distribution in Structural Steel.
3. To validate temperature distribution experimentally.
4. To predict the Distortion and Residual stress in structural Steel.
5. To study of the effect of heat input on residual stress and distortion of PTA welded plates.

1.3 LITERARTURE REVIEW

MOVING HEAT SOURCES

Heat generation in welding is based on the concept of instantaneous heat sources. The heat source model developed by Goldak is used popularly, which distributes the heat throughout the volume of the molten zone. The Goldak heat source model, shown in Figure 2 is defined spatially by a double ellipsoid. The front half of the source is the quadrant of one ellipsoidal source, and the rear half is the quadrant of a second ellipsoidal source. The power density distribution is assumed to be Gaussian along the weld path, or the z-axis on the work piece. It is convenient to introduce a coordinate, □ , fixed on the heat source and moving with it. The moving reference frame on the heat source is related to the coordinate fixes on the work piece by

$$\xi = z + \upsilon(\tau - t) \qquad (1)$$

Where □ is the welding speed, τ is a lag time necessary to define the position of heat source at t = 0. In the double ellipsoid model, the fractions of heat deposited in the front and rear of heat source are denoted by ff and fr respectively, and these fractions are specified to satisfy ff + fr = 2. Let q denote the power density in W/m3 within the ellipsoid, and let a, b, and c. denote the semi-axes of the ellipsoid parallel to the axes. Then the power density distribution inside the front quadrant is specified by[4]

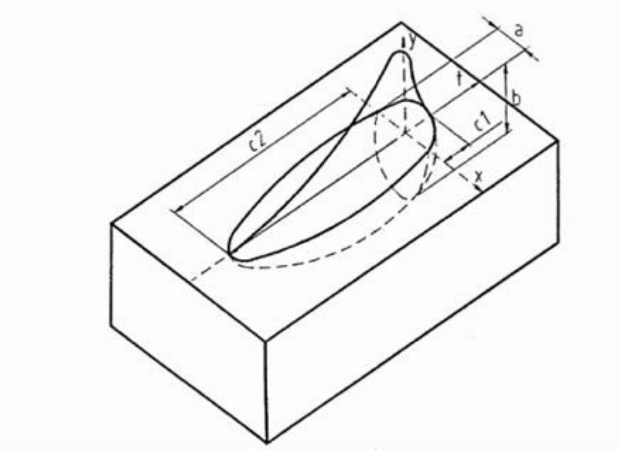


FIGURE 2. Goldak Double Ellipsoid Heat Source Model(2)

Where ff denotes fraction of heat deposited in front of heat source. a, b, c denotes the semi axes of the ellipsoid parallel to the axes. Q is the heat available at the source.

The power density in the rear quadrant is specified by

$$q(x,y,z,t) = \left(\frac{6\sqrt{3}f_f Q}{abc_1\pi\sqrt{\pi}}\right)e^{-3(x^2/a^2)}e^{-3(y\ /b^2)}e^{-3(\xi^2/c_1^2)} \qquad (2)$$

$$q(x,y,z,t) = \left(\frac{6\sqrt{3}f_r Q}{abc_2\pi\sqrt{\pi}}\right)e^{-3(x^2/a^2)}e^{-3(y\ /b^2)}e^{-3(\xi^2/c_2^2)} \qquad (3)$$

Where fr denotes fraction of heat deposited in rear of heat source. a, b, c denotes the semi axes of the ellipsoid parallel to the axes. Q is the heat available at th Q is the heat available at the source. For an electric arc the heat available is

$$Q = \eta VI \qquad (4)$$

Where η is the heat source efficiency, V, is the arc voltage, and I is the arc current. The parameters a, b, c1, and c2 are independent, and can take on different values for the front and rear quadrants of the source to properly model the weld arc.

II. ASSUMPTIONS IN FINITE ELEMENT SIMULATION

While developing three dimensional finite element model for Butt joint, following assumptions are made:

- The initial temperature of work piece is kept at 30°C
- Convection is applied on all surfaces except at bottom surface.
- Combined radiation and convection are considered by the Vinokurov’s empirical relationship.
- Forced convection due to shielding gas flow is not considered.

FINITE ELEMENT ANALYSIS (FEA) Procedure

The general finite element modeling procedure consists of the following steps.[1]

Preprocessing
- Element type definition
- Material properties definition
- Building solid model
- Meshing the solid model

Solution
- Defining initial conditions
- Applying boundary conditions
- Applying loads
- Solving for results

Post Processing
- Reading result file
- Viewing results

PRE PROCESSING

SELECTION OF ELEMENT TYPE

The size and type of the element are crucial to obtain accuracy of the result and reduction of solution time required for finite element analysis. The element chosen for the analysis is SOLID 70.e source.[3,8]

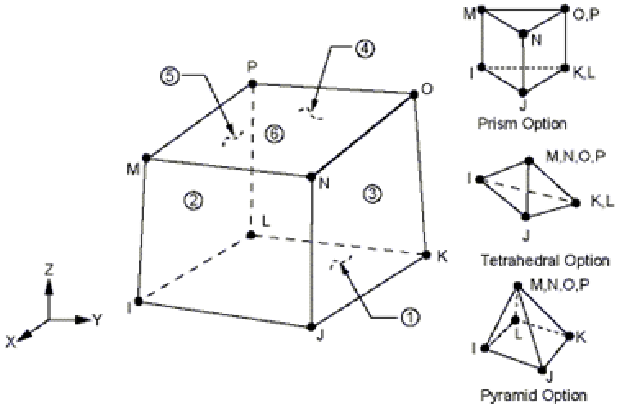


Figure 3. Details of eight noded brick element (Solid 70)

III. SPECIFIED WELD ARC MODEL PARAMETERS

The Goldak heat source model, is used to simulate the weld arc. The heat source parameters are shown in Figure 2.1, while the values of parameters are tabulated in Table 1.

The value of the heat input is calculated by using equation 4

V- Voltage in volts

I-Current in Ampere

η –Arc efficiency = 60%

Q- Heat input, W

Q = 100 * 20 * 0.6

=1200 W

Table 1. Parameters of Double Ellipsoid Heat Source Model and Calculated Heat Input

S.No.	Goldag Parameters				Heat Input Q,W
	a , mm	b , mm	C1, mm	C2, mm	
1	2	1	1.5	3	1000
2	1.5	2	2	3.5	1200
3	1.75	3	2.5	4	1500

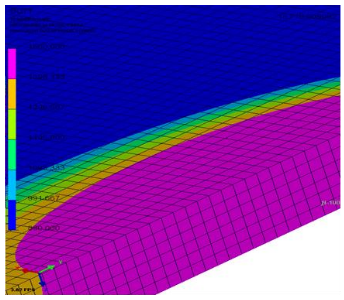


Figure 4. Temperature Contour Plot of PTA welded Structural Steel plate at Heat Input of 1000 W in SYSWELD

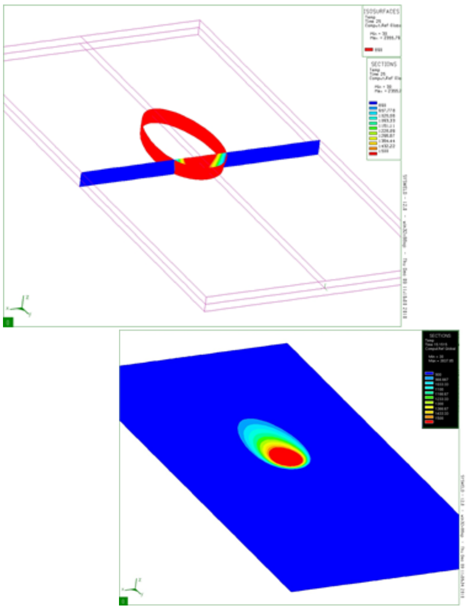


Figure 5. Temperature Contour of PTA Welded Steel Plate at the Heat Input 1200 W at 50 mm in SYSWELD

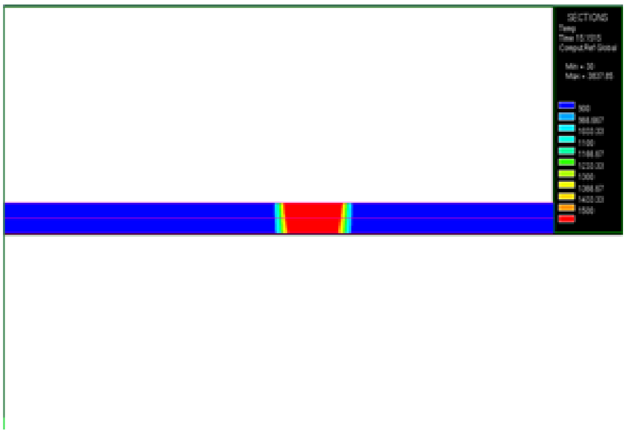


Figure 6. Temperature Contours of PTA Welded Steel Plate for the Heat Input 1500 W at 50 mm in

IV. SYSWELD

In order to simulate the thermal analysis in ANSYS, it requires storage space of 7GB and 2.5 hours for analysis in 3 GB ram and 3 GHz personal computer. When the heat input is 1000 W, the maximum temperature obtained is 2000 deg C. The further the analysis is performed for the thermal, when the heat input is 1200 W the maximum temperature obtained is 2200 deg C and when the heat input is 1500 W the maximum temperature obtained is 2413 deg C. The temperature distribution at the Node 5344 is 1500deg C.[6]

V. VALIDATION

The K type thermocouple was used for measuring temperature at a particular point when two Structural Steel Plates are welded and the results were compared with those of finite element method. The thermocouples were located in drilled holes in the work pieces. Temperatures were measured at two different points. The dimensions of Structural Steel plates are 100 mm X 50 mm X 3 mm. The voltage signals received from thermo couple are provided to a PC through Data Acquisition card manufactured by National Instruments. Lab View Software was used to display Time History plots.[4]

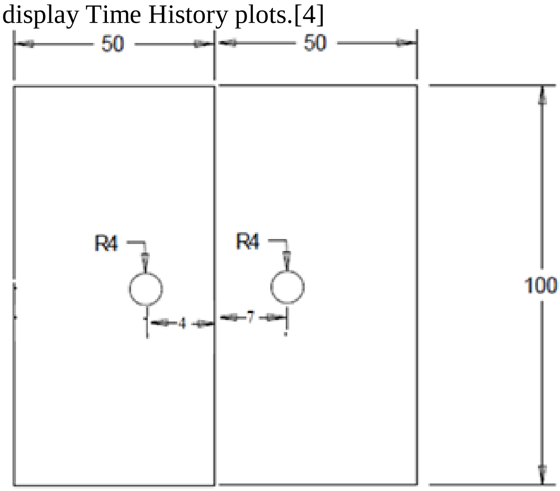


Figure 7. Locations of thermocouples in weld plate

Figure 10. Locations of thermocouples in weld plate

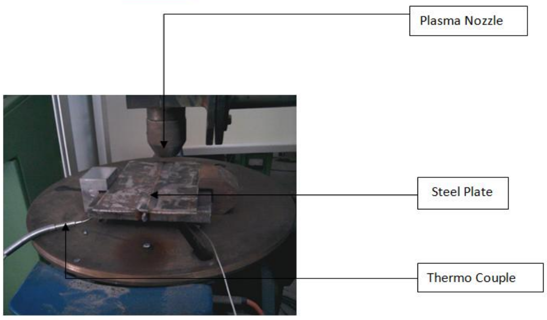
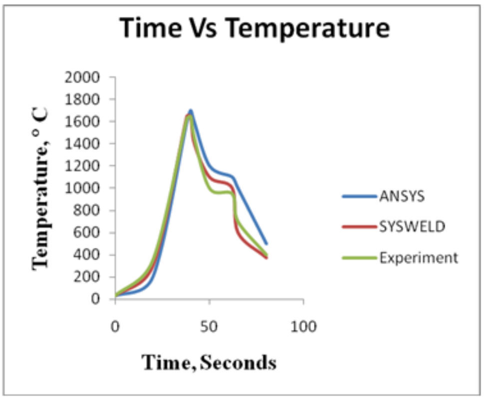


Figure 8. Experimental Setup showing Plasma Transferred Arc Welding of Steel Plates and Measurement of Temperature Using K-Type Thermocouples.



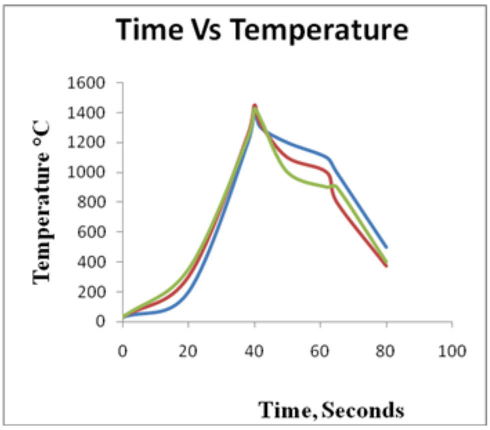


Figure 9. Temperature history for the Heat input 1200 W at a point 4 mm and 7 mm away from the Weld centre line respectively

From above figure it is evident that the predicted temperature has good agreement with that of the measured values. The maximum temperature obtained in ANSYS is 1420 deg C and in SYSWELD is around 1420 deg C. The corresponding measured temperature is 1390 deg C. This has good agreement with the simulation at the 7024th node in ANSYS and at 20000th node in SYSWELD. From the simulation, it clearly found that when the heat input increases the maximum temperature also increases.[9]

FINITE ELEMENT SIMULATION OF RESIDUAL STRESSES AND DISSTORTION IN BUTT JOINT

Table1. Temperature Dependent Mechanical Properties of Structural Steel

Temperature °C	Coefficient of Thermal Expansion α (10^{-5} K^{-1})	Poisson's ratio	Young's Modulus (10^{10} N/m^2)
70	76	0.318	7.75
140	2.04	0.32	7.69
210	2.16	0.323	7.42
280	2.25	0.325	7.23
350	2.3	0.328	6.93
420	2.37	0.333	6.62
490	2.41	0.334	5.85
560	2.49	0.335	4.93
630	2.57	0.335	3.98
700	2.65	0.331	2.78

VI. RESULTS AND DISCUSSIONS

Maximum residual stress for Heat Input of 1200 W is predicted to be 513 N/mm2 and distortion is found out to be 0.16mm.

Maximum residual Stress for Heat Input of 1500 W is predicted to be 543 N/mm2 and distortion is found out to be 0.23mm.

It is found that maximum residual stress is 491 N/mm2 and the distortion is found to be 0.189 mm in ANSYS

with increase in the heat input, distortion increases non linearly. Upon cooling after welding, the weld pool solidifies and shrinks, exerting stresses on the surrounding weld metal and HAZ. The stresses produced from thermal expansion and contraction exceed the yield strength of the parent metal, localized plastic deformation of the metal occurs. Plastic deformation results in lasting change in the component dimensions and distorts the structure. This causes distortion of weldments.[7]

VII. CONCLUSION

A three dimensional finite element model was developed to predict the temperature distribution, residual stresses and distortion.

- A 3-D non-linear transient thermal model was developed to predict the temperature distribution using the concept of Finite Element Method.
- The process was simulated using ANSYS 12 Software and ANSYS Parametric Design Language (APDL) code was developed for the same.
- The process was also simulated using SYSWELD.
- Maximum temperature during the experiment was found to be 1650 □ C and 1420 □ C.
- The developed model was validated with the experimental results and a good agreement was found.
- Structural analysis were made and was able to predict the residual stress and distotion.
- Various Parameters of goldag equation gives the various bead shape in the base metal.
- Increase in the amount of heat input increses the residual stress and distortion.

REFERENCES

[1] Geng J, Tan KBC, Liu GJ. Application of finite element analysis in implant dentistry: A review of the literature.Prosthet Dent 2001; June 585-98

[2] Kazuo Tanne, Mamoru Sakuda, Charles J. Burstone. Three dimensional finite analyses for stress distribution in the periodontal tissue by orthodontic forces. Am J Orthod Dentofacial Orthop1987; Volume 92(6):499-505

[3] N. Brännberg, J. Mackerle, (1994) "Finite element methods and material processing technology", Engineering Computations, Vol. 11 Iss: 5, pp.413 - 455

[4] W.G. Cochran, G.M. Cox, Experimental Designs, John Willy & sons Singapore 1957

[5] Marimuthu and Murugan Effects of Plasma Transferred Arc Welding parameters on welding bead geometry, in computational **materials** science and surface engineering vol. 8 pp 51-88 (2006)

[6] R.Kaul,P.Ganesh,S.K.Albert,A.Jaishwal, N.P. Lalla ,A. Gupta C.P.Pal ,A.K.NathLasercladdingof austenitic stainless steel with nickel base hard facing alloy, Surface Engineering19 (2003)pp269-273.

[7]. Gatto, A., Bassoli, E., Fornari, M. Plasma Transferred Arc Deposition of Powdered High Performances Alloys: Process Parameters Optimisation as a Function of Alloy and Geometrical Configuration Surface and Coatings Technology 187 2004: pp. 265 – 271.

[8]. Cochran, W.G and G.M. Cox, 1962. Experimental Design. Asia Publishing House, Bombay, India

[9]. Harris, P. and B.L. Smith, 1983. Factorial technique for weld quality prediction. Metal Constr., 15(11): 661-666

Dr. P V Senthiil

Ph.D. (Manufacturing Simulation)
National Board Expert member
Prof & Head Mechanical Sciences
St.Peter's University
Mobile: +91 9842074506
Email: headmechspu@gmail.com
Per: ***drpvsenth@gmail.com***

Professional / Academic background:

Dr. P.V Senthiil is currently working as Prof & Head, Mechanical Engg at St Peter's University,Chennai, India. He lectures modules in CNC machines, Rapid Prototyping and AI & Robotics. He is also guiding 10 researchers in different disciplines.

Dr. Senthiil’s work experience as a Factory consultant / knowledge transfer partnership associate working for electronic goods consumer industry in south Africa, France, UK. He was involved in developing, implementing & documenting operating procedures in advanced applications CNC industries for inventory reduction activities within various sectors of the company.

Prior to start of his research, Dr Senthiil was working as a Professor at Coimbatore Institute of Technology between May 1986 to Jun 2011. He did his masters in Mechanical Engineering at PSG Tech, April 1985, Bachelor’s at PSG Tech, in Mechanical Engineering, April 1983.

Deputed by Tamil nadu government to Norway University, Asmara,East Africa for one year during 2005

Expert member of NBA for the last 7 years
Anna University Affilation Committee member from 2009

Life member of Indian Society of Technical Education, Institute of Supply Chain Management and Institute ofManagement Consultancy

Editorial Advisory board member “The RPD Magazine”, a bi-monthly magazine published in India.

Editorial board member for TES and Emerald International journal publishers (UK)

Editorial board member for International Journal of Scheduling journal publishers

WSAC (World Science Academy Council Member

Visiting Professor – Nottingham University, MALAYSIA

Visiting Professor – AIT, THAILAND