Strength Analysis And Optimization of Orbital Welding Parameters

Ashish Saxena [1], Prof. Vivek Babele [2]
[2] HOD, Dept of Mechanical
[1, 2] Rajeev Gandhi Proudyogiki Mahavidyalaya Bhopal

Abstract- *Orbital welding was first used in the 1960's when the aerospace industry recognized the need for a superior joining technique for aerospace hydraulic lines. In this pipe welding method, the welding head rotates around a fixed vertical or horizontal pipe. As this process is fully automatic it provides better weld quality and greater surface finish. The construction of weld head is not complicated so it can be move from one place to another easily and because of portable weld head orbital welding is used in many industries. It provides more precision and reliable method than the manual welding method. Orbital welding is specifically used for joining of tubes but now-a-days it is also used in joining of tube-tube and tube to sheet.*

In this research FEM is applied. In stage-I, 3D model is prepared with orbital welding boundary conditions and orbital pipe welding process is simulated using ANSYS software to predict the temperature distribution and residual stresses in the butt welded pipes. Using three process parameters with three levels, design of experiment is performed in Mini-Tab software to get 27 dominating experimental sets. Orbital Pipe welding is performed using values of these 27 sets. Temperature during the welding process is recorded as output parameter. Optimization of process parameters is done using TAGUCHI, ANOVA, and RESPONSE SURFACE METHOD through Mini-Tab software. The thesis end with comparison of theses experimental Temperature values with the simulated values obtained from stage to validate the simulated model.

Keywords- FEM, FEA, ANOVA, TAGUCHI, RESPONSE SURFACE METHOD, MIG Welding, DOE, Medium Carbon Steel.

I. INTRODUCTION

Context

Orbital welding was first used in the 1960's when the aerospace industry recognized the need for a superior joining technique for aerospace hydraulic lines. In this pipe welding method, the welding head rotates around a fixed vertical or horizontal pipe. As this process is fully automatic it provides better weld quality and greater surface finish. The construction of weld head is not complicated so it can be move from one place to another easily and because of portable weld head orbital welding is used in many industries. It provides more precision and reliable method than the manual welding method. Orbital welding is specifically used for joining of tubes but now-a-days it is also used in joining of tube-tube and tube to sheet.

Tungsten Inert Gas Welding

Tungsten inert gas (TIG) welding is also called as gas tungsten arc welding (GTAW). Orbital welding uses principle of TIG welding. TIG welding is an inert-gas shielded arc welding process which uses a non-consumable electrode. In TIG welding heat is derived from arc between anode and cathode. It comes under the category of fusion welding process where the joint is completely melted to obtain the fusion. Inert gas is used to protect the molten weld from atmospheric contamination. The setup and welding torch are shown in figures 1.1 and 1.2.

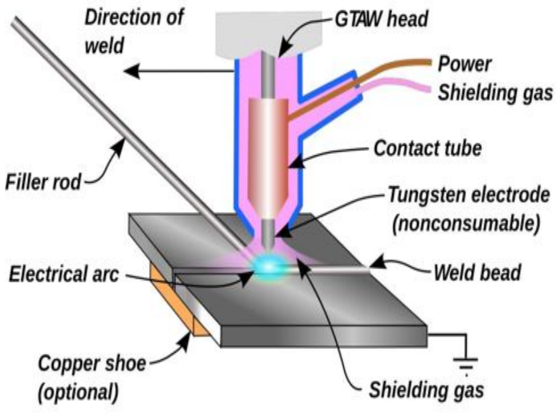


Figure 1: TIG welding setup

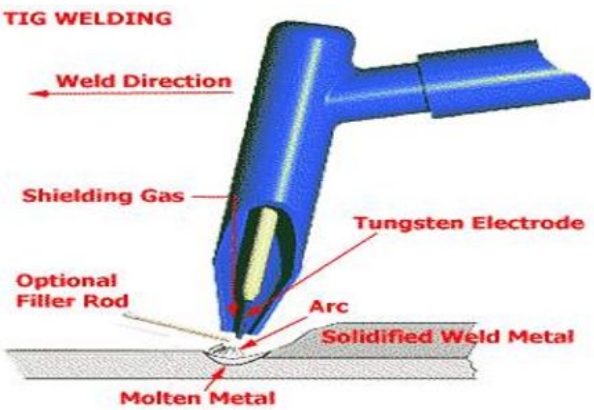


Figure 2: TIG welding torch

Welding Torch

TIG welding torch are generally rated on the basis of current carrying capacity because the welding speed depends on current carrying capacity. TIG welding torch can be divided into.

- Air cooled
- Water cooled

Air cooled is generally used for lower current range then water cooled.

Electrode

A non-consumable electrode is used in TIG welding to perform the necessary joint. The electrode used contains 1 to 2% thoria (thorium oxide) mixed along with the core tungsten or tungsten with 0.15% to 0.40% zirconia (zirconium oxide). The tungsten electrodes are less expensive but will carry less current. Throated tungsten electrodes carry high currents and maintain a stable arc. Materials like stainless steel, aluminum, motel, cast iron uses throated tungsten electrode in TIG welding process. Figure 1.3 shows the geometry of TIG electrode, while figure 1.4 shows the TIG electrodes for AC and DC TIG welding.

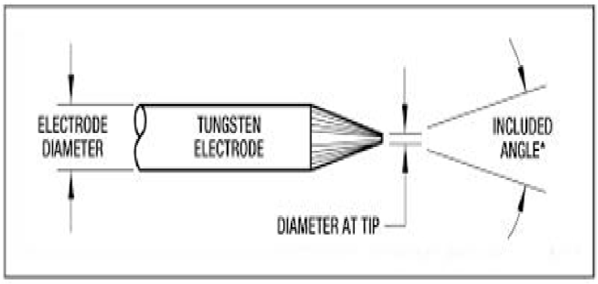


Figure 3: TIG Electrode

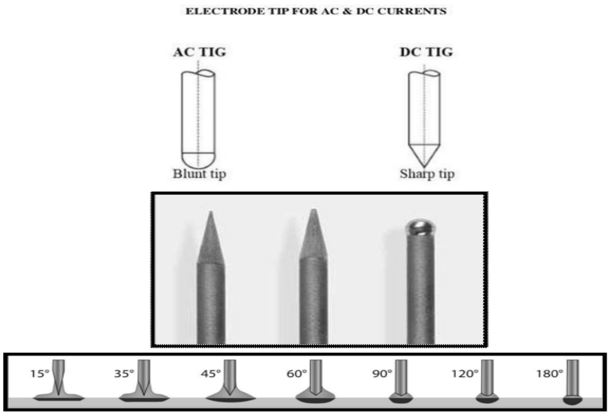


Figure 4: Various TIG Electrodes

METHODOLOGY

Steps to Be Followed

To perform investigations, the adopted methodology comprises of following steps:

Activity 1: Identification of area of research
Activity 2: Literature review
Activity 3: Formulation of research problem
Activity 4: Formulation of objectives
Activity 5: Study of optimization techniques.
Activity 6: Study of process and performance parameters of pipe Orbital welding method.
Activity 7: Selection of pipe, process and performance parameters related to pipe Orbital welding method.
Activity 8: Design of experiments
Activity 9: Experimentation using selected orbital pipe welding parameters.
Activity 10: Optimization of Orbital pipe welding parameters.
Activity 11: Study of exiting FE models to simulate Orbital pipe welding and selection of best model.
Activity 12: Integrating optimized parameters to develop FE model of an Orbital welded component.
Activity 13: Prediction of residual stresses and tensile strength of welded pipe.
Activity 14: Results analysis and report preparation.

Simulation of Orbital Welding Using ANSYS

In this topic of thesis we discuss the orbital welding simulation in two stages these are:

Finite Element Analysis: In this stage, 3D model is developed and Electro Thermal analysis of orbital welding is to be carried out using ANSYS software to predict residual stresses and temperature distribution in welded pipes.

Finite Element Method: Finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems with the help of differential equations. It uses variation methods (the calculus of variations) to minimize an error function and produce a stable solution. Analogous to the idea that connecting many tiny straight lines can approximate a larger circle, FEM encompasses all the methods for connecting many simple element equations over many small subdomains, named finite elements, to approximate a more complex equation over a larger domain.

In engineering problems there are some basic unknowns. If they are found, the behavior of the entire structure can be predicted. The basic unknowns are the Field variables which encountered in the engineering problems. In a continuum, these unknowns are infinite. The finite element procedure reduces such unknowns to a finite number by dividing the solution region into small parts called elements and by expressing the unknown field variables in terms of assumed approximating functions (Interpolating functions/Shape functions) within each element. The approximating functions are defined in terms of field variables of specified points called nodes or nodal points. Thus in the finite element analysis the unknowns are the field variables of the nodal points. Once these are found the field variables at any point can be found by using interpolation functions. After selecting elements and nodal unknowns next step in finite element analysis is to assemble element properties for each element. For example, in solid mechanics, we have to find the force-displacement i.e. stiffness characteristics of each individual element. Mathematically this relationship is of the form

$$[k]_e \{\delta\}_e = \{F\}_e$$

Where $[k]_e$ is element stiffness matrix, $\{\delta\}_e$ is nodal displacement vector of the element and $\{F\}_e$ is nodal force vector. The element of stiffness matrix k_{ij} represent the force in coordinates direction „i" due to a unit displacement in coordinate direction „j". Four methods are available for formulating these element properties viz. direct approach, variation approach, weighted residual approach and energy balance approach. Any one of these methods can be used for assembling element properties. In solid mechanics variation approach is commonly employed to assemble stiffness matrix and nodal force vector. Element properties are used to assemble global properties/structure properties to get system equations;

$$[k]\{\delta\} = \{F\}.$$

Then the boundary conditions are imposed. The solution of these simultaneous equations gives the nodal unknowns.

Using these nodal values additional calculations are made to get the required values e.g. stresses, strains, moments, etc. in solid mechanics problems.

Basic Steps of Finite Element Method

- Select suitable field variable and the type of element.
- Discretize the continua.
- Select interpolation function.
- Find the element properties.
- Assemble element properties to get global properties.
- Impose the boundary conditions.
- Solve the system equations to get the nodal unknowns.
- Make the additional calculation to get the required values.

METHODOLOGY

To perform investigations, the adopted methodology comprises of following steps:

Activity 1: Identification of area of research
Activity 2: Literature review
Activity 3: Formulation of research problem
Activity 4: Formulation of objectives
Activity 5: Study of optimization techniques.
Activity 6: Study of process and performance parameters of pipe Orbital welding method.
Activity 7: Selection of pipe, process and performance parameters related to pipe Orbital welding method.
Activity 8: Design of experiments
Activity 9: Experimentation using selected orbital pipe welding parameters.
Activity 10: Optimization of Orbital pipe welding parameters.
Activity 11: Study of exiting FE models to simulate Orbital pipe welding and selection of best model.
Activity 12: Integrating optimized parameters to develop FE model of an Orbital welded component.
Activity 13: Prediction of residual stresses and tensile strength of welded pipe.
Activity 14: Results analysis and report preparation.

Simulation of Orbital Welding Using ANSYS

In this topic of thesis we discuss the orbital welding simulation in two stages these are:

Finite Element Analysis

In this stage, 3D model is developed and Electro Thermal analysis of orbital welding is to be carried out using ANSYS software to predict residual stresses and temperature distribution in welded pipes.

Finite Element Method

Finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems with the help of differential equations. It uses variation methods (the calculus of variations) to minimize an error function and produce a stable solution. Analogous to the idea that connecting many tiny straight lines can approximate a larger circle, FEM encompasses all the methods for connecting many simple element equations over many small subdomains, named finite elements, to approximate a more complex equation over a larger domain.

In engineering problems there are some basic unknowns. If they are found, the behavior of the entire structure can be predicted. The basic unknowns are the Field variables which encountered in the engineering problems. In a continuum, these unknowns are infinite. The finite element procedure reduces such unknowns to a finite number by dividing the solution region into small parts called elements and by expressing the unknown field variables in terms of assumed approximating functions (Interpolating functions/Shape functions) within each element. The approximating functions are defined in terms of field variables of specified points called nodes or nodal points. Thus in the finite element analysis the unknowns are the field variables of the nodal points. Once these are found the field variables at any point can be found by using interpolation functions. After selecting elements and nodal unknowns next step in finite element analysis is to assemble element properties for each element. For example, in solid mechanics, we have to find the force-displacement i.e. stiffness characteristics of each individual element. Mathematically this relationship is of the form

$$[k]_e \{\delta\}_e = \{F\}_e$$

Where $[k]_e$ is element stiffness matrix, $\{\delta\}_e$ is nodal displacement vector of the element and $\{F\}_e$ is nodal force vector. The element of stiffness matrix k_{ij} represent the force in coordinates direction „i‟ due to a unit displacement in coordinate direction „j‟. Four methods are available for formulating these element properties viz. direct approach, variation approach, weighted residual approach and energy balance approach. Any one of these methods can be used for assembling element properties. In solid mechanics variation approach is commonly employed to assemble stiffness matrix and nodal force vector. Element properties are used to assemble global properties/structure properties to get system equations;

$$[k] \{\delta\} = \{F\}.$$

Then the boundary conditions are imposed. The solution of these simultaneous equations gives the nodal unknowns.
Using these nodal values additional calculations are made to get the required values e.g. stresses, strains, moments, etc. in solid mechanics problems

Basic Steps of Finite Element Method

- Select suitable field variable and the type of element.
- Discretize the continua.
- Select interpolation function.
- Find the element properties.
- Assemble element properties to get global properties.
- Impose the boundary conditions.
- Solve the system equations to get the nodal unknowns.
- Make the additional calculation to get the required values.

II. EXPERIMENTAL AND NUMERICAL STUDIES

Design of Experiments

The experimental runs are performed based on the L_{27} orthogonal array of TAGUCHI method. On the basis of literature reviewed and industrial feedback suitable process parameters and performance parameters are selected, experimentation is done using the closed head orbital welding set up. After completing the 27 experiments tensile test is performed on all the 27 welded pieces to find out the tensile strength of the welded pipe. All the experimental values are shown in table.

Table: Experimental results based on L_{27} orthogonal array

Experiment No.	Orthogonal array L_{27}			Output Parameters	
	Process parameters			Experimental Temperature	Tensile Strength (MPa)
	Welding current (amp)	Voltage (Volt)	Gas flow rate (m/sec)		
1	100	10	9	1479	732
2	100	10	11	1106	825
3	100	10	14	987	953
4	100	15	9	792	1005
5	100	15	11	1461	791
6	100	15	14	1085	848
7	100	20	9	2674	575
8	175	20	11	1991	602
9	175	20	14	1061	967
10	175	10	9	2350	532
11	175	10	11	1627	594
12	175	10	14	1125	882
13	175	15	9	2705	683
14	175	15	11	2767	512
15	175	15	14	1421	721
16	175	20	9	4686	464
17	175	20	11	3408	486
18	175	20	14	1964	581
19	250	10	9	3461	493
20	250	10	11	2625	523
21	250	10	14	1326	836
22	250	15	9	5138	374
23	250	15	11	2980	480
24	250	15	14	2302	568
25	250	20	9	6895	321
26	250	20	11	4985	468
27	250	20	14	2964	506

UTM machine specification

The machine used to test the specimen is shown in figure 4.4.
Design – UTM 200
Capacity – 200KN
Maximum tensile clearance – 50-700
Distance between columns – 500
Piston stroke – 200
Maximum straining speed – 150 mm/min
Weight - 1400 Kgs

Figure : Universal testing machine used to determine the tensile strength of the welded pipe specimen

Selected pipe material

AISI 304 stainless steel is used as a work piece material for the present experiment. The chemical composition of AISI 304 stainless steel is presented in table 4.4. All the 27 pipes on which experimentation done are shown in below in fig 4.5.

Table : Chemical composition of AISI 304 stainless steel

Material	C	Si	Cr	Mn	Fe	Ni	Tb
AISI 304 stainless steel	3.28	3.34	14.89	1.07	53.54	5.2	18.6

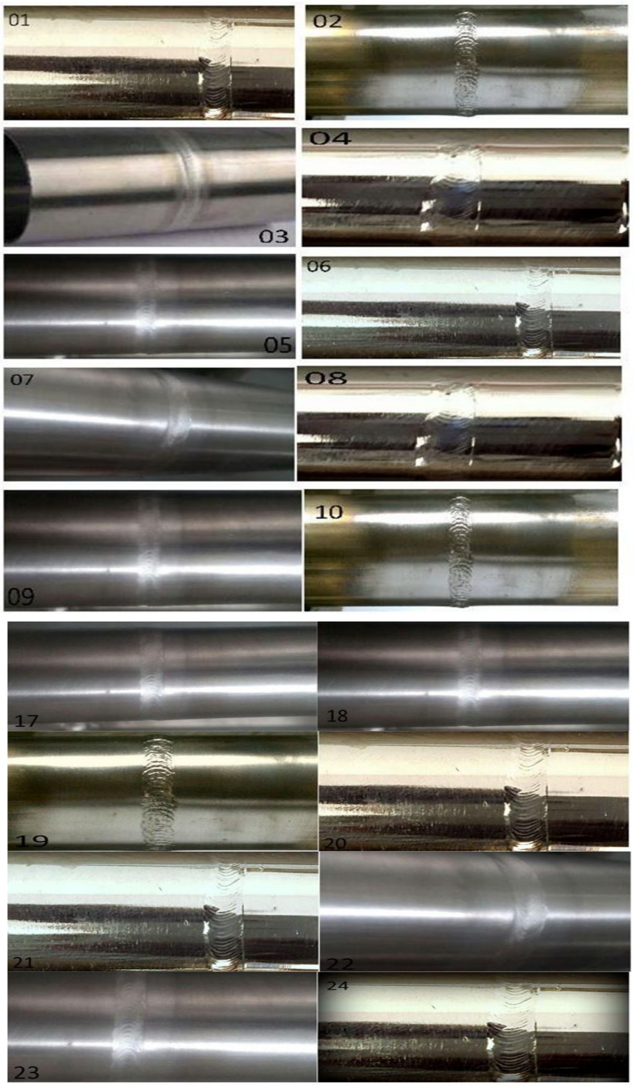


Figure : Experimentation done on AISI 304 stainless steel

Estimation of tensile strength & temperature through experimentation

The orbital welding experiments are conducted using Taguchi, orthogonal array, ANOVA. The effects of individual orbital welding process parameters, on the selected quality characteristics – current, voltage, gas flow rate, welding speed etc. are calculated separately below. The average value and S/N ratio of the response characteristics for each variable at different levels are calculated from experimental data. The main effects of process variables both for raw data and S/N data are plotted. The main effects plots are used for examining the effects of parameters on the optimization characteristics. The analysis of variance (ANOVA) of raw data and S/N data is carried out to identify the significant variables and to quantify their effects on the response characteristics. The (optimal settings) of process variables in terms of mean response characteristics are established by analyzing the response curves and the ANOVA tables.

Table : S/N Ratio of temperature per each experiment

Experiment no	Temperature	S/N ratio
1	1479	63.3994
2	1106	61.6269
3	967	60.5633
4	792	59.9302
5	1461	62.6772
6	1083	61.4597
7	2674	64.9791
8	1991	65.3338
9	1041	60.3490
10	2350	66.6433
11	1627	63.6768
12	1125	61.0231
13	3705	71.3738
14	2747	69.0877
15	1421	63.0519
16	4686	72.4360
17	3408	70.3913
18	1964	65.3628
19	3461	70.2670
20	2625	67.7656
21	1326	61.7698
22	5138	74.1650
23	3980	71.9977
24	2302	66.4327
25	6895	76.5280
26	4985	73.8128
27	2964	69.1395

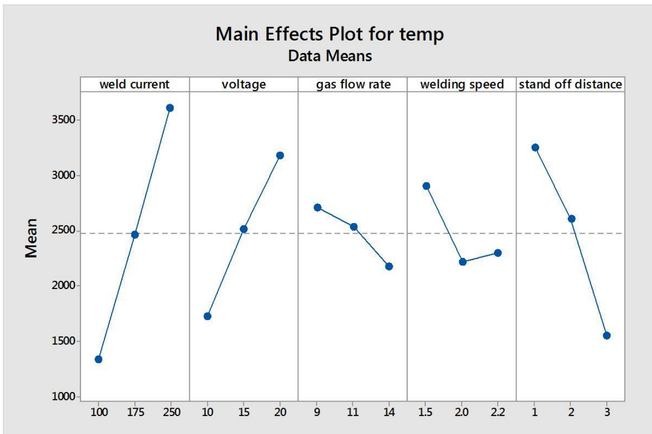


Figure : Effects of process parameters on temperature (raw data)

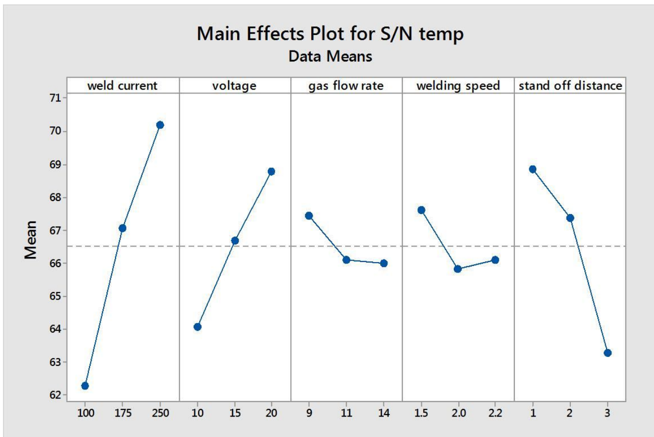


Figure: Effects of process parameters on temperature (S/N data)

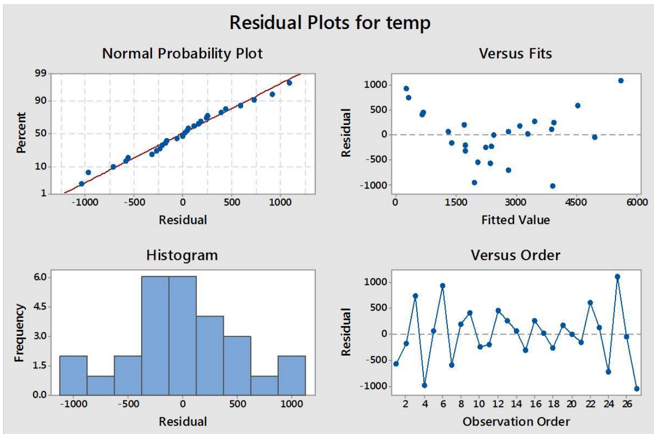


Figure : Residual plots for temperature (raw data)

Table : Analysis of variance for temperature

Source	DOF	Sum Of Square	Mean Square	F Value	p-value Prob > F
A-current	2	287.05	143.52	48.22	0.017
B-voltage	2	99.76	49.881	16.76	0.007
C-gas flow rate	2	11.34	5.670	1.90	0.851
D-welding speed	2	16.71	8.35	2.81	0.585
E-standoff Distance	2	149.64	74.82	25.14	0.506
Residual Error	16	47.63	2.977		
Total	26	612.14			
S = 1.72534		R-Sq = 92.22%		R-Sq(adj) = 87.36%	

Simulated Study

In Two-Stage FEA, first the orbital welding process is simulated to predict the residual stresses and temperature distribution in the welded pipes while in second stage tensile test on theses welded pipes is simulated to predict the tensile strength of the joint. In Stage-I 3D FEM is used while in Stage-II 2D Axi-symmetric Finite Element approach is adopted.

Calculation of Gaussian Heat Flux : To start the simulated study electrical input (Voltage, Current) should be converted into thermal input. To achieve this Gaussian heat distribution is assumed and calculated using following equation for 27 selected experimental sets.

$$Q(r) = \frac{\eta I e^{-(r^2/2\sigma^2)}}{2\pi\sigma^2}$$

Where $Q(r)$ = surface flux at radius r, η = efficiency coefficient/ Arc efficiency V = voltage, I = current, σ = radial distance from center

Finite Element Analysis

4.3.2.1 3D simulated environment for Orbital Pipe Welding Process CAD Modelling:

It helps to create the 3D geometry of the part/assembly of which you want to perform FEA.

Dimensions of 3D model:
Length of hollow pipe – 0.5m
Inner diameter of pipe – 0.045m
Outer diameter of pipe – 0.042m
Thickness – 0.003m

Figure : Diagram of CAD Line Model

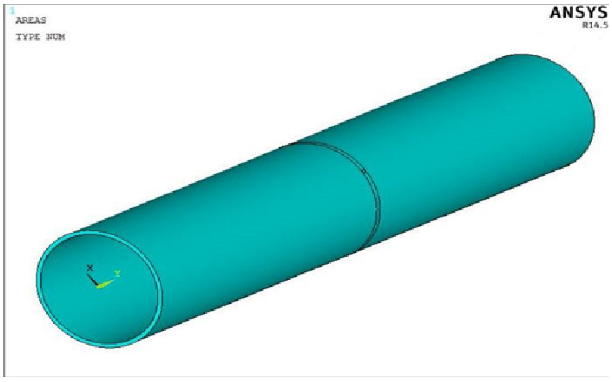


Figure : 3D cad model

Meshing

- Meshing is a critical operation in FEA. In this operation, the CAD geometry is divided into large numbers of small pieces. The small pieces are called mesh. With the increase in mesh size, the finite element analysis speed increase but the accuracy decrease.
- Meshing follows a number of steps like element selection, defining material property, mesh size to generate mesh in cad model.

Mesh type: hexahedral mesh

Element type: 8 node brick 45. Solid45 is used for 3D modelling of solid structure.

Table . gives the Material properties used in cad model to perform FE simulation.

AISI 304 Stainless steel	
Thermal properties	**Mechanical properties**
Conductivity -17 w/m-k	Young"s modulus-190Gpa
Specific heat – 490j/kg-k	Poission"s ratio – 0.29
Density-7.85mg/m3	
Thermal expansion-16e-6 per k	

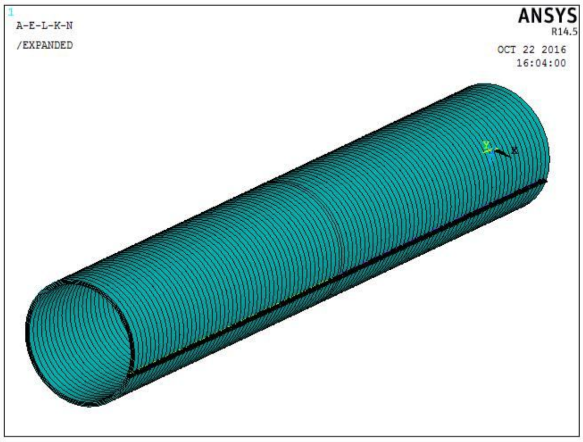


Figure : Meshing part

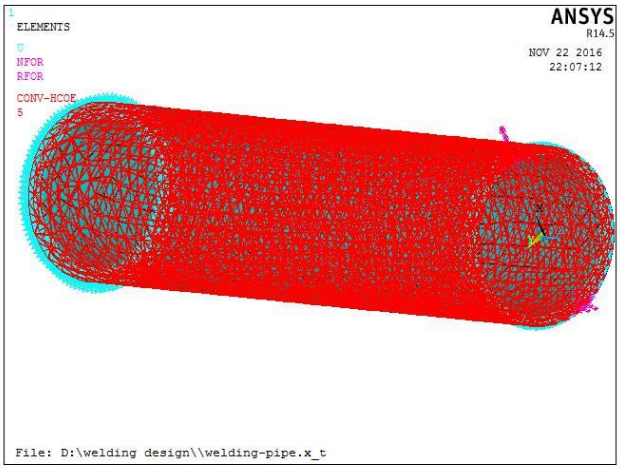


Figure : Load and boundary condition

Post processing: viewing the results

After defining the geometry, type of element, mesh size, boundary condition and solving and post processing has been performed to get temperature and residual stresses for all the 27 cases which are tabulated in Tables 4.2 and 4.5. Table 4.10 gives the simulated temperature value and residual stress value by applying Gaussian heat flux value.

Table : Post processing results

Exp.no	ANSYS temperature (K)	Analytical Heat flux (J/mm)	ANSYS residual stress (Mpa)
1	1388.1	566	250
2	1033.2	425	992
3	823.5	386	537
4	715.7	850	474
5	1537.4	637.5	535
6	900.04	579.5	575

7	2751.04	1133	1500
8	2041.5	850	1810
9	1191.72	772.7	709
10	2410.2	991.6	1320
11	1789.4	743.7	1610
12	1045.8	676.13	648
13	3602.9	1487.6	1950
14	2671.6	1115.7	2320
15	1556.32	1014.3	908
16	4795.5	1983.3	3010
17	3553.8	1487.5	2600
18	2066.7	1352.2	1150
19	3432.5	1416.66	1180
20	2545.6	1062.5	1020
21	1483.4	965.9	625
22	5136.32	2125	2790
23	3805.9	1593.7	1820
24	2212.6	1448.86	1220
25	6840.1	2833	5490
26	5066.26	2125	1020
27	2941.8	1931.8	1620

After getting the simulated residual stress values by applying Gaussian heat distribution, few samples of residual stress are plotted in figure 4.15. Similarly simulated temperature distribution samples are plotted in 4.16.

Case 1

Current	100 A
Voltage	10 V
Gas Flow Rate	9
Speed	1.5 meter per second
Stand Off Distance	2 mm
Max. Residual Stress	992 Mpa
Min. Residual Stress	110 Mpa

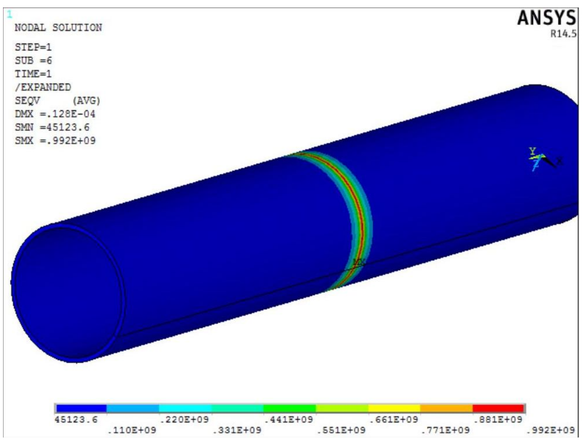


Figure : Residual stress plot from ANSYS

Case-2

Current	100 A
Voltage	10 V
Gas Flow Rate	9
Speed	1.5 meter per second
Stand Off Distance	3 mm
Max. Residual Stress	164 Mpa
Min. Residual Stress	537 Mpa

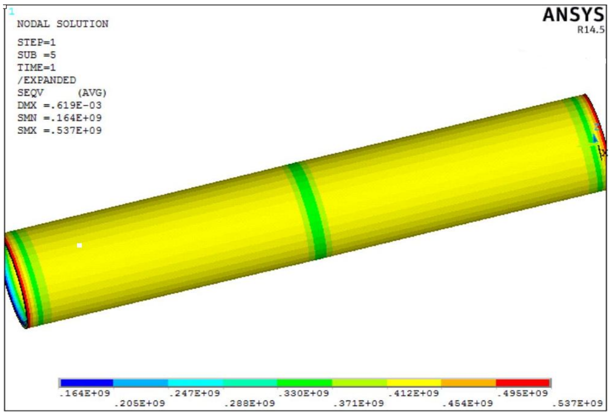


Figure : Simulated Temperature Distribution Samples (Case-2)

Case-3

Current	100 A
Voltage	15 V
Gas Flow Rate	11
Speed	2 meter per second
Stand Off Distance	1 mm
Max. Residual Stress	137 Mpa
Min. Residual Stress	447 Mpa

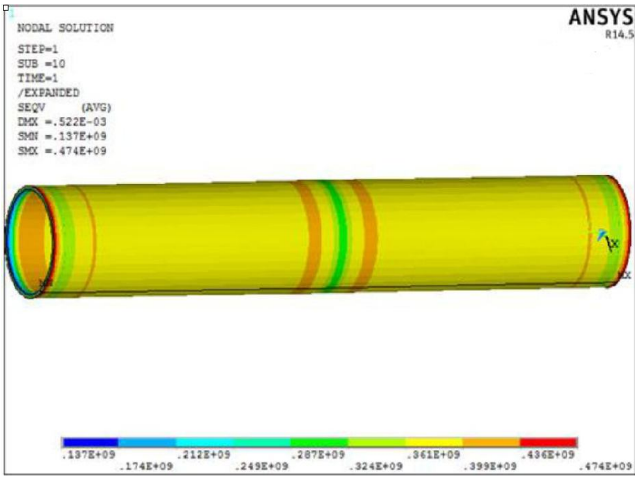


Figure : Simulated Temperature Distribution Samples (Case-3)

Case 4

Current	100 A
Voltage	15 V
Gas Flow Rate	11
Speed	2 meter per second
Stand Off Distance	3 mm
Max. Residual Stress	675 Mpa
Min. Residual Stress	180 Mpa

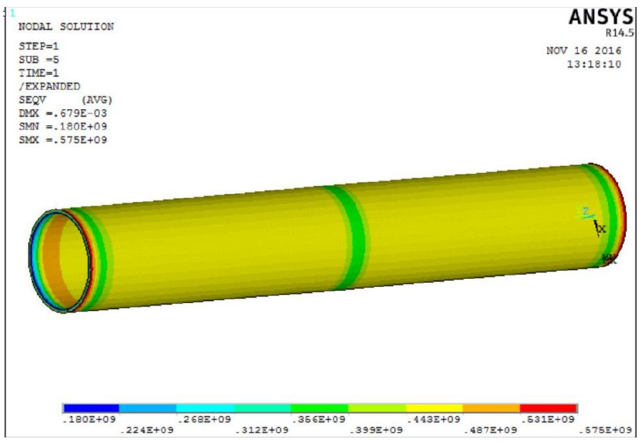


Figure : Simulated Temperature Distribution Samples (Case-4)

Validation of Simulated Model

As seen from table 4.10 the simulated temperatures are in match with the experimental temperature values and all the simulated temperature values are within the expected range of 1200-2000 except few odd cases. This comparison is the basis of validation of our numerical model of Orbital welding prepared using ANSYS platform. On comparing temperature of both experimentation and numerical value percentage error is develop which is shown in table 4.11

Table comparing temperature of both experimentation and numerical value percentage error is develop

Exp. No.	ANSYS temperature (K)	Experimental Temperature (K)	Percentage Error
1	1388.1	1479	6.15
2	1033.2	1106	6.66
3	823.5	867	5.07
4	715.7	792	9.72
5	1537.4	1461	-5.20
6	900.04	1083	16.89
7	2751.04	2674	-2.8
8	2041.5	1991	-2.5
9	1191.72	1041	-14.4
10	2410.2	2350	-2.5
11	1789.4	1767	-1.2
12	1045.8	1125	7.1
13	3602.9	3705	2.7
14	2671.6	2747	2.7
15	1556.32	1421	-9.5
16	4795.5	4686	-2.32
17	3553.8	3508	-1.2
18	2066.7	1964	-5.1
19	3432.5	3461	0.83
20	2543.6	2625	3.04
21	1483.4	1326	-11.84
22	5136.32	5138	0.3
23	3805.9	3980	4.3
24	2212.6	2302	3.9
25	6840.1	6895	0.79
26	5066.26	4985	-1.62
27	2941.8	2964	0.7

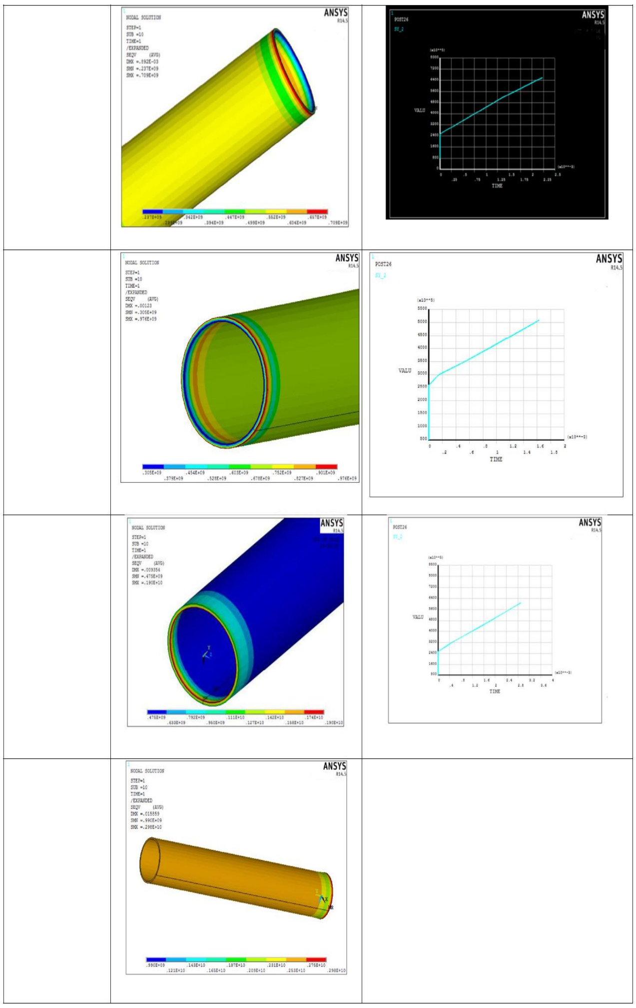
Finite Element Analysis:

Simulation of Tensile Test of welded pipes

A two dimensional model is used with residual stresses of previous stage to simulate tensile test on the welded pipe so as to predict the tensile strength.

Table : Material properties used in 2d cad model
AISI 304 Stainless steel

Thermal properties	Mechanical properties
Conductivity -17 w/m-k	Young"s modulus-190Gpa
Specific heat – 490j/kg-k	Poission"s ratio – 0.29
Density-7.85mg/m3	Yield strength -240mpa
Thermal expansion-16e-6 per k	



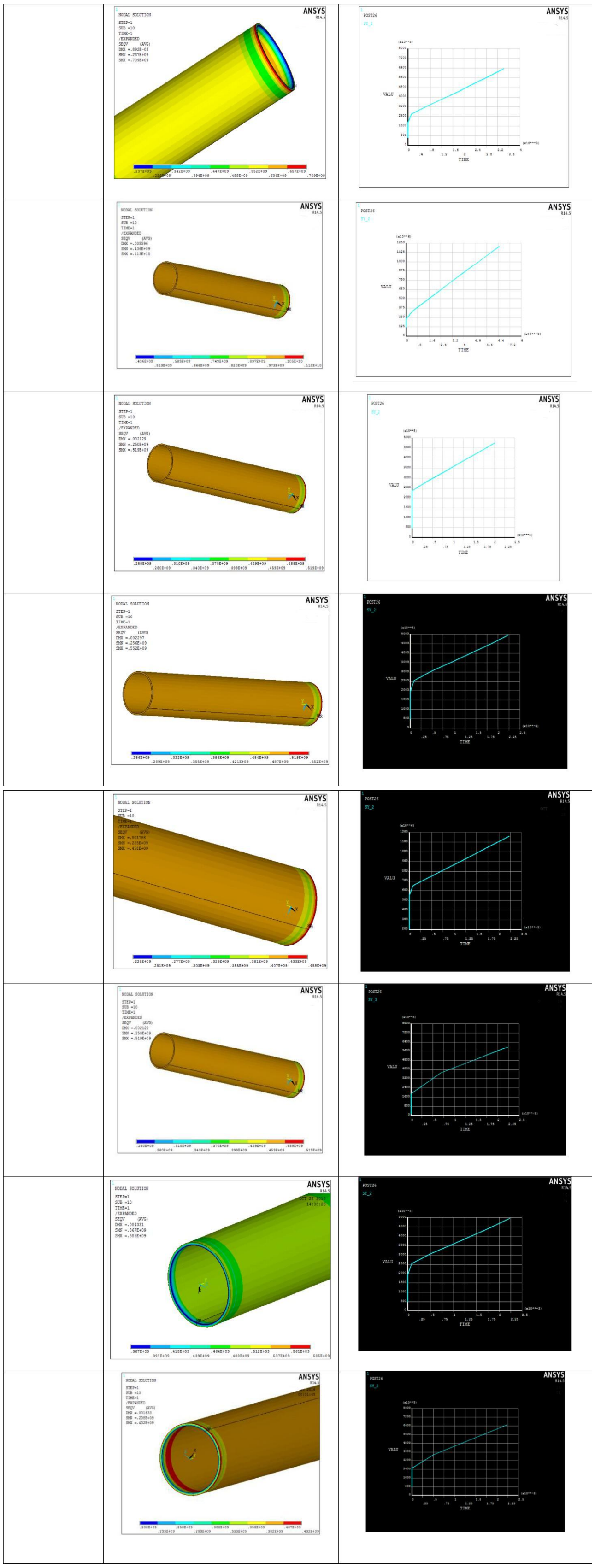


Figure : showing tensile strength plot using ANSYS

Table : Comparison of tensile strength experimental

Exp. No.	Experimental tensile strength	Simulated tensile strength	% Error
1	732	848	13
2	825	976	12
3	953	1000	4
4	3005	2980	0.08
5	791	709	10.1
6	848	1130	21.0
7	575	519	10.2
8	602	552	8.3
9	967	872	10.2
10	532	538	-1.11
11	564	562	0.35
12	882	911	-3.1
13	463	458	1.07
14	512	519	1.3
15	721	585	18.6
16	464	432	6.89
17	486	467	3.9
18	581	552	4.8
19	493	479	2.83
20	523	526	0.5
21	836	782	6.4
22	374	371	0.82
23	480	455	5.2
24	568	548	3.5
25	321	313	2.5
26	448	383	14.5
27	504	497	1.38

III. FUTURE SCOPE

This work will be further investigations and optimization of the process parameters during Orbital pipe welding of different material of pipes. Analysis of Variance (ANOVA) has been employed to see the level of significance of each process parameter on performance measures. And also create the different geometry and used different approach for investigate and optimize the stress and strength of the orbital welded material.

REFERENCES

[1] N.I. S. Hussein, M. N. Ayof, S. Nordin [2016] "Tensile Strength of Orbital Welded Mild Steel Tubes with Dissimilar Thickness" International Journal of Materials, Mechanics and Manufacturing.

[2] Bhawan deep singh, Avtar simgh [2015] "Performance of activated TIG process in mild steel welds", e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 12, Issue 2 Ver. IV (Mar - Apr.), PP 01-05,www.sciencedirect.com.

[3] Abhishek B P, Anil kumar G, Madhusudhan.T [2015] "Experimental And Finite Element Analysis of Thermally Induced Residual Stresses for Stainless Steel 303 grade Using Gmaw Process", International Research Journal of Engineering and Technology (IRJET), Volume: 02 Issue: 02 | May.

[4] Dinesh Kumar. R, Elangovan.S, Siva Shanmugam. N [2014] "Parametric Optimization Of Pulsed – TIG Welding Process In Butt Joining Of 304l Austenitic Stainless Steel Sheets", IJRET: International Journal of Research in Engineering and Technology ISSN: 2319-1163 | pISSN: 2321-7308, Volume: 03 Special Issue: 11 | NCAMESHE.

[5] M. Islama, A. Buijk, M. Rais-Rohani, K. Motoyama [2014] "Simulation-Based Numerical Optimization Of Arc Welding Process For Reduced Distortion In Welded Structures", 0168-874X & Elsevier B.V.

[6] Paulo Roberto de Freitas Teixeira1, Douglas Bezerra de Araújo, Luiz Antônio Bragança da Cunha [2014] "Study Of The Gaussian Distribution Heat Source Model Applied To Numerical Thermal Simulations Of Tig Welding Processes", Ciência & Engenharia (Science & Engineering Journal ISSN 1983-4071 23 (1): 115 – 122, jan. – jun., 115.

[7] Vishnu V.S, Nadeera M, Joy Varghese V.M [2014] "Numerical Analysis of Effect of Process Parameters on Residual Stress in a Double Side TIG Welded Low Carbon Steel Plate", International Conference on Advances in Engineering & Technology – 2014 (ICAET).

[8] Shankar Sehgal, Harmesh Kumar [2014] "Structural Dynamic Finite Element Model Updating using Derringer‟s Function: A Novel Technique", E-ISSN: 2224-3429 11 Volume.

[9] A. Ravisankar, Satish Kumar Velaga, Gaurav Rajput, S. Venugopal, 2014, "Influence of welding speed and power on residual stress during gas tungsten arc welding (GTAW) of thin sections with constant heat input: A study using numerical simulation and experimental validation", Journal of Manufacturing Processes, Volume 16, Issue 2, April 2014, Pages 200-211.

[10] Olivier Desmaison, Gildas Guillemot, Michel Bellet [2013] "Numerical Modelling Of Hybrid Arc/laser

Welding: A Level Set Approach For Weld Bead Formation and Residual Stresses", Proc. JOM17, Int. Conf. on Joining Materials, May 5-8, Helsingor, Denmark, International Institute of Welding

[11] Pablo Batista Guimarães, Paulo Marcelo Almeida Pedrosa, Yogendra Prasad Yadava, José Maria Andrade Barbosa, Aníbal Veras Siqueira Filho, Ricardo Artur Sanguinetti Ferreira [2013] " Determination of Residual Stresses Numerically Obtained in ASTM AH36 Steel Welded by TIG Process" Materials Sciences and Applications, 2013, 4, 268-274.

[12] Dheeraj Singh, Vedansh Chaturvedi, JyotiVimal [2013] "Parametric optimization of TIG process parameters using Taguchi and Grey Taguchi analysis" Issue 3, Vol.4 (June-July 2013) http://www.rspublication.com/ijeted/ijeted_index.htm ISSN 2249-6149.

[13] M. A. Daha, G. A. Nassef, I. A. Abdallah, [2012] "Numerical Modeling of Heat Transfer and Fluid Flow In Keyhole Plasma Arc Welding Of Dissimilar Steel Joints", ISSN: 0975-5462 Vol. 4 No.02.