International Journal of Engineering and Technical Research (IJETR)
ISSN: 2321-0869, Volume-02, Issue-02, February 2014

Finite Element Simulation of Resistance Welding of High Density Polyethylene Pipe

Sachin Kumar, Yash Kumar, Amandeep Singh

***Abstract*— High density poly ethylene (HDPE) has been leading material for gas and water pipelines due to its many advantageous properties over metals such as lower weight, higher chemical and corrosion resistance. The paper has investigated the resistance welding process of a high density polyethylene which has been experimented under three operating parameters, namely current, pressure and time. The present research is dedicated exclusively to finite element simulation of resistance welding of HDPE pipes. Joining of the two HDPE pipes carried out by resistance welding process. The software used for the FE analysis is ANSYS. In the FEA main aim is to make a finite element model and simulation. At first thermal analysis is carried out according to the parameters current and time by considering variation of material properties occurring at different levels of temperature and then structural analysis done for the values of pressure. The study is carried out how the temperature is distributing along the axial direction and with respective to the time is found out and then different structural deformation i.e. strain, material loss and weld bead height is found out.**

***Index Terms*— HDPE, Resistance Welding, Finite Element Analysis**

I. Introduction

Resistance welding is a fusion welding process that requires the application of both heat and pressure to achieve a sound joint. The simplest form of the process is spot welding where the pressure is provided by clamping two or more overlapping sheets between two electrodes. A current is then passed between the electrodes, sufficient heat being generated at the interface by resistance to the flow of the current that melting occurs, a weld nugget is formed and an autogenously fusion weld is made between the plates. The heat generated depends upon the current, the time the current is passed and the resistance at the interface. The resistance is a function of the resistivity and surface condition of the parent material, the size, shape and material of the electrodes and the pressure applied by the electrodes.

HDPE is a common plastic that can be heat welded in the field using specialized equipment. Fish farmers may find this material useful when used for building hauling tanks, fish tanks, or other farming related uses. As a plastic, HDPE material is lighter than metal, non-corrosive, long lasting, and is a poor conductor of electrical current. High density polyethylene (HDPE) is a hard plastic that is used in many applications due to its' resistance to degradation and fatigue. Some of the commercial products made from this material include drainage culverts, pipes, and whitewater kayaks. There are specialty companies that can custom make just about any product needed, however the average person can quickly learn how to properly prepare and weld this material.

Manuscript received Feb. 13, 2014.

Welding of polyethylene materials gives new possibilities of applications due to the fact that both materials have significantly different physical, chemical and mechanical properties. These thermoplastic materials have good elevated temperature resistance, strength and wear which helps to use it for many fields of electronic, aerospace, nuclear and automotive industries. Friction welding is a solid state welding technology with good quality and reliability. In friction welding process, the temperature field, the stress field and the strain field are the three important factors, which directly affect the quality of welded joint. Resistance welding is considered to be the most significant development in metal joining in a decade and is a "green" technology due to its energy efficiency, environment friendliness, and versatility.

A. Finite Element Analysis

a) FEA model and mesh

FEA model of RSW process is shown in Fig 3 and 4, which is axisymmetric about y axis since only half portion of the complete model is analysed. The x axis is the contact surface of the two sheets called as faying surface. The model is meshed using three elements; PLANE223, CONTA172 AND TARGE169. The element PLANE223 with structuralthermoelectric capabilities has eight nodes with up to four degrees of freedom per node. It has UX, UY, TEMP and VOLT degrees of freedom. The other elements are contact elements consisting of contact pair of CONTA172 and TARGE169. Contact occurs when the element surface penetrates one of the target segment elements (TARGE169) on a specified target surface. Any translational or rotational displacement, forces, moments, temperature, voltage and magnetic potential can be imposed on the target segment element. [6] [7]

b) Material models and welding conditions

Temperature varying properties are considered for copper electrode and mild steel sheets. The properties assigned are thermal conductivity, resistivity, Young's modulus, coefficient of thermal expansion, yield stress, specific heat and contact resistivity. These temperature dependent properties are assigned in the range of 21ºC to 1204ºC [4] . In modelling RSW process with the complicated thermoelectric behavior, several physical phenomena must be considered. It is of great importance to define the parameters correctly to obtain correct results. Since electric current has great influence on the quality of RSW process, the input current in this simulation is 50 Hz sine wave AC current of 10 kA, applied for 200 ms. The current is imposed as an electric load on the top surface of upper electrode. To simulate the cooling process of the welding, the hold time is taken as 60 ms.

II. METHODS

A. SELECTION OF MATERIAL

For simulation of the Resistance Welding (RW)

experiments, pure HDPE are used for specimen are in the size 40mm internal diameter ,60mm external diameter and length of 60mm. Stainless steel mesh of 150 microns having 40mm internal diameter and 60mm external diameter is used. Terminals of resistance welding machine are connected to this mesh during welding process.

B. SELECTION OF PROCESS PARAMETERS

The Factors Varied for the current experiment are Current, Time and Pressure. The Levels for Parameters, Current and Time are determined by subjecting mesh to different current and analysing the temperature of the mesh at different time intervals.

C. DESIGN OF EXPERIMENTS

The design technique helps us study many factors (variables) simultaneously and most economically. By studying the effects of individual factors on the results, the best factor combination can be determined. Full factorial design is used to carry out experiments, in which responses are measured at all combinations of the factor levels.

D. FINITE ELEMENT SIMULATION

The finite element simulation of the Resistance welding process is done as explained below. ANSYS which is a widely used software used for Finite Element simulation is used here. To get good simulation results, first we have to consider the basic material properties, here HDPE. Then modelling of experiment can be done accordingly. The welding process is a non-linear transient state. The thermal physical properties of the material changes rapidly with the change of temperature. The controlling equation of heat conduction is:

$$\rho c\frac{\partial T}{\partial t}=\frac{\partial}{\partial x}(\lambda\frac{\partial T}{\partial x})+\frac{\partial}{\partial y}(\lambda\frac{\partial T}{\partial y})+\frac{\partial}{\partial z}(\lambda\frac{\partial T}{\partial z})+Q \quad \text{......................} \quad (1)$$

In the equation, ρ, c and λ are the density of the material, the specific thermal capacity of the material and the heat conductance of material respectively as functions of temperature and Q is the intensity of inner heat source.

III. OBSERVATIONS

A. Material Properties

Finite element simulation of resistance welding is a nonlinear analysis. The material properties of hdpe material are changing with temperature.The properties at different temperature used in this study are shown in Table 1.

Table1. Properties of HDPE at different temperatures

S. no	Temperature (k)	Thermal conductivity (W/mK)	Specific heat (J/Kg K)	Density (Kg/ m^3)	Young's modulus MPa	Thermal expansion(10 e-5)/K
1	300	.46	1500	956	1000	9.5
2	320	.44	1600	946	850	9.5
3	340	.42	1700	934	600	9.5
4	360	.40	1800	922	380	14.2
5	380	.38	1900	904	200	14.2
6	400	.36	2000	842	100	18.0

B. Process Parameters

The Levels for Parameters, Current and Time are determined by subjecting mesh to different current and analysing the temperature of the mesh at different time intervals. Parameters used in each level are as shown in Table 2.

Table 2. Process parameters with their values at 3 levels

	Pressure (Kg/cm^2)	Current (A)	Time(s)
Leve1 1	4	20	5
Level 2	6	25	10
Level 3	8	30	15

C. Design of Experiments

The design technique helps us study many factors (variables) simultaneously and most economically. By studying the effects of individual factors on the results, the best factor combination can be determined. Full factorial design is used to carry out experiments, in which responses are measured at all combinations of the factor levels which are as shown in Table 3.

Table 3. Process parameters in different levels as per simulation run order

S.no	Current(A)	Pressure(kg/cm^2)	Time(s)
1	**20**	**4**	**5**
2	20	4	10
3	20	4	15
4	20	6	5
5	20	6	10
6	20	6	15
7	20	8	5
8	20	8	10
9	20	8	15

10	25	4	5
11	25	4	10
12	25	4	15
13	25	6	5
14	25	6	10
15	25	6	15
16	25	8	5
17	25	8	10
18	25	8	15
19	30	4	5
20	30	4	10
21	30	4	15
22	30	6	5
23	30	6	10
24	30	6	15
25	30	8	5
26	30	8	10
27	30	8	15

Once the factors are assigned to the columns, individual experiments are easily described. The individual experiments are represented by the rows, which can be described in plain language.

D. *Finite Element Simulation*

The finite element simulation of the Resistance welding process is done as explained below. ANSYS which is a widely used software used for Finite Element Simulation is used here.

E. *Creating the model:*

Model of HDPE specimen and stainless steel mesh is created by using the modelling software of CATIA.The solid 70 (Brick 8 node 70) elements is adopted for the model.

a) Creating the geometry

The whole model is divided into three parts and modelled separately. Two models of HDPE with dimensions 40mm internal diameter, 60mm external diameter and length of 60mm dimension form the two work pieces to be welded together. Creating the geometry is as shown in Figure 1. The middle element which is considered as the mesh part, heating section in which heat is generated is placed in between 2 HDPE specimen elements. After creating three parts separately, three parts assembled into the one part in CATIA.

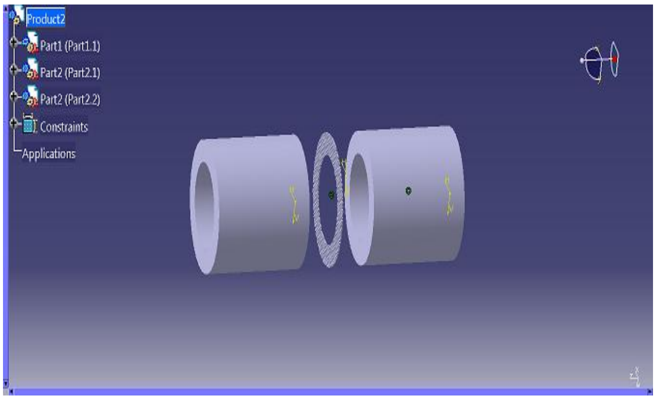

Fig 1. Arrangement for RW specimen

b) SOLID70 Element description

SOLID70 has a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. SOLID70 Geometry is shown in Figure 2.

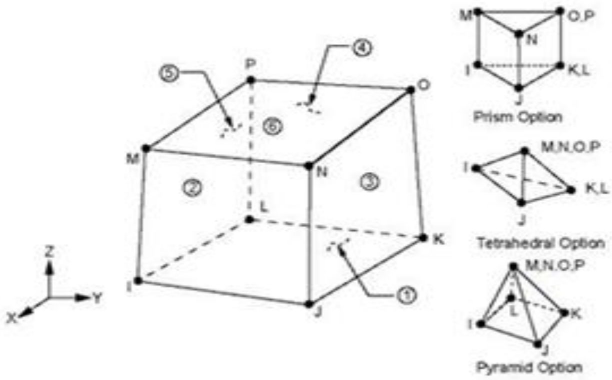

Figure 2. SOLID70 Geometry

c) Material properties and meshing

After creating specimen in CATIA, import the solid geometry in ANSYS. Initially we have to do thermal analysis after that that thermal results file is the input to the structural analysis. A free mesh, which has no restriction in terms of element shapes, and has no specified pattern applied to it has be used here with automatic meshing tool is used with a fineness level of 7/10.The meshed model is shown in Figure 3.

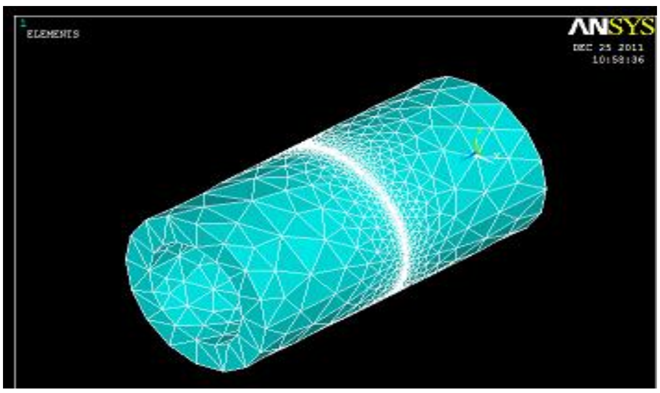

Fig 3. Meshed model

d) Thermal analysis

Stainless steel mesh of 150microns used with dimensions 40mm internal diameter ,60mm external diameter and thickness 0.3mm. In this mesh has 120 threads which are the carriers of electricity and thus heat generated will depend only on this 120 threads. Welding machine used (TORNADO 201) will supply constant output voltage of 63 V as per the manual and has an efficiency of 85%.So total power produced by mesh in the specified volume can be calculated as:

P = (V x I x 0.85 x 120) / (0.3e-3)................(2)

As the experiment based on transient analysis, time is another important parameter, which can be directly fed into the programme as time sub step. The heat generated can be given applied to the defined interval and finally results obtained. Table 4 shows the heat generated for 3 levels of current used.

Table 4. Heat generated according to the current input

Current(A)	Heat generation (J)
20	428400000
25	535500000
30	642600000

e) Structural analysis

Visualising the deformation occurred and obtaining stress, strain and displacement values in , Z axis can be obtained by structural analysis. Applying pressure on the areas in ANSYS and deformation comes in the shape after applying the pressure as shown in figure 4 & 5.

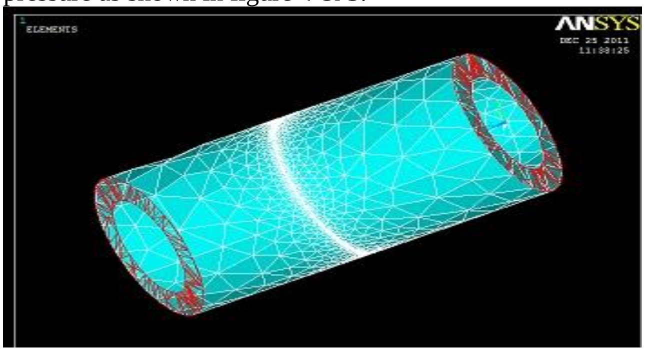

Fig 4. Pressure applied as visualised

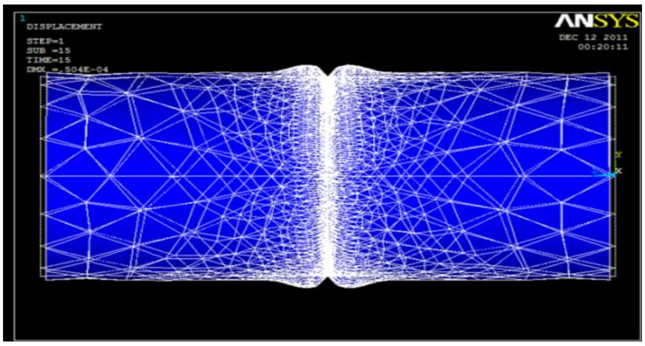

Fig 5. Shows deformed shape after apply the pressure

f) FEM Simulated Results

Design of experiment gives the list of experiments which has to be performed so as to obtain the results with the help of finte element modelling. The results simulated by using finite element modelling as shown in Table 5.

Table 5. FEM simulated results

S.no	Current(A)	Pressure (Kg/cm2)	Time(S)	Temp(K)	Strain Z-axis	Bead height(mm)	Metal loss (10-4m)	Bead width mm	Heat effected width(mm)
1	20	4	5	369	.0117	.51	2.12	6.6	8.8
2	20	4	10	404	.0158	.94	2.95	7.9	12.8
3	20	4	15	432	.0201	1.2	3.72	9.4	14
4	20	6	5	369	.0100	.73	2.75	7.3	8.8
5	20	6	10	404	.0178	1.1	3.52	8.6	12.8
6	20	6	15	432	.0280	1.43	4.23	10.2	14
7	20	8	5	369	.0130	.90	3.45	8.1	8.8
8	20	8	10	404	.0168	1.5	4.12	9.3	12.8
9	20	8	15	432	.024	1.74	4.98	11.1	14
10	25	4	5	387	.0138	.83	3.12	8.9	9.6
11	25	4	10	431	.0237	1.32	4.1	10.1	13.8
12	25	4	15	468	.0342	1.49	4.82	12.2	14.6
13	25	6	5	387	.0129	1.22	3.82	9.5	9.6
14	25	6	10	431	.0279	1.54	4.6	10.3	13.8
15	25	6	15	468	.0352	1.63	5.32	12.1	14.6
16	25	8	5	387	.0132	1.45	4.32	9.9	9.6
17	25	8	10	431	.0221	1.73	5.2	10.3	13.8
18	25	8	15	468	.033	1.97	6.01	12.9	14.6
19	30	4	5	405	.0129	1.27	3.81	10.2	11.4
20	30	4	10	459	.0308	1.53	4.68	11.2	14.8
21	30	4	15	505	.0325	1.82	5.43	13.4	16
22	30	6	5	405	.0222	1.49	4.34	10.8	11.4
23	30	6	10	459	.0336	1.72	5.29	11.2	14.8
24	30	6	15	505	.0453	2.3	5.9	12.6	16
25	30	8	5	405	.016	1.92	4.98	11.5	11.4
26	30	8	10	459	.040	2.21	5.97	12.4	14.8
27	30	8	15	505	.047	3.02	6.45	14.8	16

IV. RESULT AND DISCUSSION

The objective of the project is to obtain the finite element model of the resistance welding specimens and do the simulation. Simulation done considering ideal conditions. Actual values may differ according to the process conditions. The temperature profile created by the software after simulation and the structural deformation occurred is visualised and simulated values are discussed.

For three levels of current values (20A, 25A, 30A) and for three levels of time (5s, 10s, 15s) the maximum temperature obtained is tabulated and plotted as shown in Figure 6. The temperatures reached corresponding each experiment number. Maximum temperature obtained at the parameters 30A and 15s.

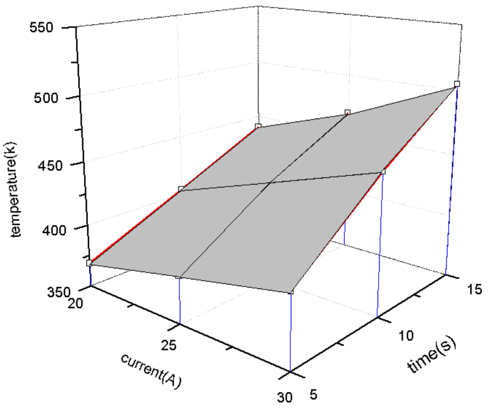

Fig. 6 Surface plot of Temperature v/s Current and Time

A. Variation of displacement

For each experiment, maximum material loss occurred along Z axis, the direction through which pressure is acted while conducting the experiment is noted and plotted.

- Material loss values obtained are tabulated and plotted keeping current constant and other two parameters, Pressure and Time varied is as shown in Figure 7.
- Material loss (Z axis) values keeping current value constant as 20A are plotted taking the variables as Pressure on X axis, Time on Y axis as in Figure 7(a)
- Material loss (Z axis) values keeping current value constant as 25A are plotted taking the variables as Pressure on X axis, Time on Y axis as in Figure 7(b).
- Material loss (Z axis) values keeping current value constant as 30A are plotted taking the variables as Pressure on X axis, Time on Y axis as in Figure 7(c).

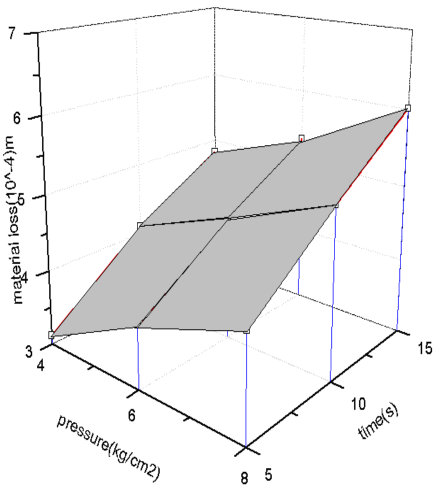

Fig. 7(a) Constant Current 20A

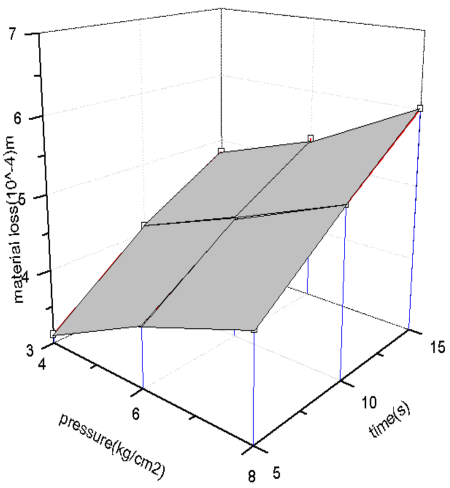

Fig. 7(b) Constant Current 25A

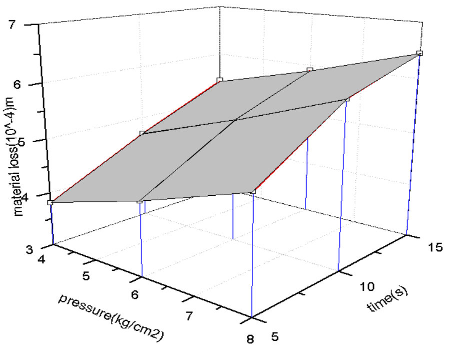

Fig. 7(c) Constant Current 30A

B. Surface plot of material loss vs current, Time

- Material loss values obtained are tabulated and plotted keeping Pressure constant and other two parameters, Current and Time varied is as shown in Figure 8.
- Material loss (Z axis) values keeping pressure value constant as 4 bar are plotted taking the variables as Current on X axis, Time on Y axis as in Figure 8(a).
- Material loss (Z axis) values keeping pressure value constant as 6 bar are plotted taking the variables as Current on X axis, Time on Y axis as in Figure 8(b).

- Material loss (Z axis) values keeping pressure value constant as 8 bar are plotted taking the variables as Current on X axis, Time on Y axis as in Figure 8(c

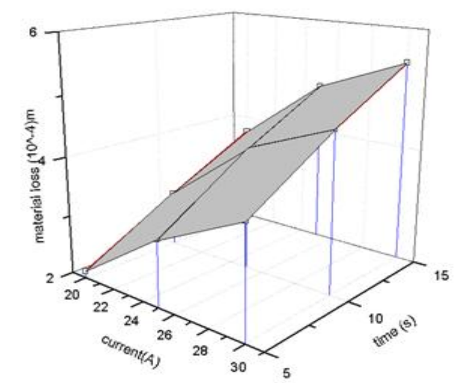

Fig. 8(a) Constant Pressure 4 kg/cm^2

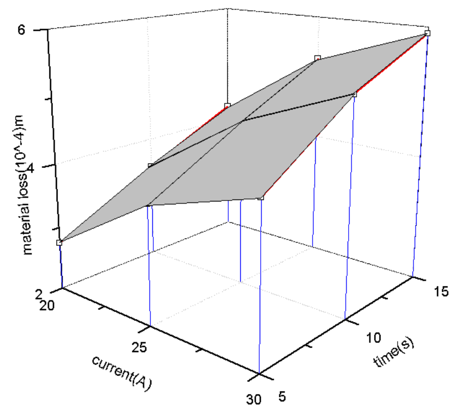

Fig. 8(b) Constant Pressure 6 kg/cm^2

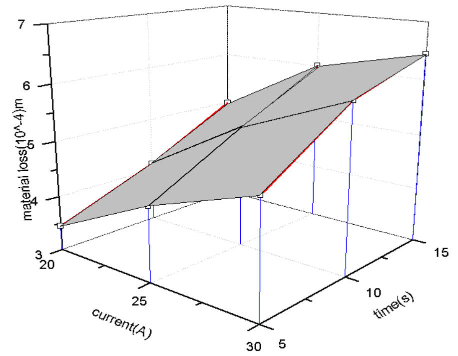

Fig. 8(c) Constant Pressure 8 kg/cm^2

C. OPTIMISATION OF PROCESS PARAMETERS

The optimal combination of the process parameters can be done by finding the signal-to-noise ratio (S/N) for each performance variable. Usually there are three categories of performance characteristics in the analysis of the S/N ratio: the lower-the –better, the higher-the-better, and the nominal-the-better

a) SIGNAL TO NOISE RATIO

Regardless of the category of the performance characteristics, a larger S/N ratio corresponds to better performance characteristics. Therefore the optimal level of the process parameters is the level with the highest S/N ratio. The signal to noise ratio is being find out by calculating the loss function (Lij). The loss function Lij of the higher the better performance characteristics can be expressed as:

$$(S/N) \text{ for MRR} = -10\log(L_{ij})$$

$$\text{Where } L_{ij} = 1/(Y_{ij})^2$$

The loss function Lij of the lower the better performance characteristics can be expressed as:

$$(S/N) \text{ for TWR AND SR} = -10\log(L_{ij})$$

$$\text{Where } L_{ij} = (Y_{ij})^2$$

Where lij is the loss function of the ith performance characteristics in the jth experiment.

b) ANOVA FOR CURRENT, PRESSURE AND TIME

From the results of ANOVA, it can be found current is the most affecting metal loss, then after time is affecting and pressure is least affecting as shown in Table 6. For minimize for metal optimum level C1,P1,T1.

Table 6.1 S/N Response table for metal loss

Level	Current (c)	Pressure(p)	Time(t)
1	**-10.73**	**-11.44**	**-10.97**
2	-13.09	-12.70	-12.87
3	-14.24	-13.92	-14.22
Delta	3.51	2.47	3.24
Rank	1	3	2

Table 6.2 Response table for mean

Level	Current	Pressure	Time
1	3.538	3.861	3.634
2	4.590	4.419	4.492
3	5.206	5.053	5.207
Delta	1.668	1.192	1.572
Rank	1	3	2

c) ANOVA FOR BEAD HEIGHT

From the results of ANOVA, it can be found current is the most affecting bead height, then after time is affecting and pressure is least affecting as shown in Table 7. For minimize for bead height optimum level C1, P1, T1.

Table 6.3 Analysis of variance (Current, Pressure, Time)

Source	df	Seq SS	Adj SS	Adj MS	F	P	Contribution (%)
Current	2	57.575	57.575	28.788	144.81	0.0	42
Pressure	2	27.561	27.561	13.781	69.32	0.0	20
Time	2	47.803	47.803	23.901	120.23	0.0	34
Error	20	3.976	3.976	.199			2.9
Total	26	136.915					

Table 7.1 Signal to Noise Ratios for bead height

Level	Current	Pressure	Time
1	**-0.4240**	**-1.1584**	**-0.5605**
2	-3.0951	-2.9257	-3.3364
3	-5.3927	-4.8277	-5.0149
Delta	4.9687	3.6693	4.4544
Rank	1	3	2

Table 7.2 Response table for mean

Level	Current	Pressure	Time
1	1.117	1.212	1.147
2	1.464	1..462	1.510
3	1.920	1.827	1.844
Delta	.803	.614	.698
Rank	1	3	2

Table 7.3 Analysis of Variance for bead height

Source	df	Seq SS	Adj SS	Adj MS	F	P	Contribution %
current	2	111.303	111.303	55.652	67.75	0.0	39
pressure	2	60.614	60.614	30.307	36.90	0.0	21
Time	2	91.094	91.094	45.547	55.45	0.0	32
Error	20	16.428	16.428	.821			5.87
Total	26	279.438					

d) ANOVA FOR BEAD WIDTH

From the results of ANOVA, it can be found current is the most affecting bead height, then after time is affecting and pressure is least affecting as shown in Table 8. For minimize for bead width optimum level C1,P1,T1

Table8.1 Signal to Noise Ratio for bead width

Level	Current	Pressure	Time
1	-18.71	-19.81	-19.15
2	-20.52	-20.14	-20.05
3	-21.54	-20.81	-21.57
Delta	2.83	1.00	2.42
Rank	1	3	2

Table 8.2 Response table for mean

Level	Current	Pressure	Time
1	8.722	9.989	9.200
2	10.689	10.289	10.144
3	12.011	11.144	12.078
Delta	3.289	1.156	2.878
Rank	1	3	2

Table 8.3 Analysis of variance for bead width

source	df	Seq SS	Adj SS	Adj MS	F	P	Contribution%
current	2	36.955	36.955	18.478	148.08	0.0	52
pressure	2	4.688	4.688	2.344	18.78	0.0	6.5
time	2	26.857	26.857	13.428	107.62	0.0	37
Error	20	2.496	2.496	0.125			3.5
Total	26	70.996					

Hence, S/N ratio for responses such as current, time, bead height and bead width has been obtained as shown in Table 9.

TABLE 9. S/N RATIO FOR RESPONSES

Stdorder	Runorder	Current	Pressure	Time	Bead height	Bead width	SNRA1	SNRA2
12	1	25	4	15	1.49	12.2	-3.4637	-21.727
23	2	30	6	10	1.72	11.2	-4.7105	-20.984
8	3	20	8	10	1.5	9.3	-3.5218	-19.3697
17	4	25	8	10	1.73	10.3	-4.76092	-20.2567
1	5	20	4	5	0.51	6.6	5.8486	-16.3909
22	6	30	6	5	1.49	10.8	-3.46373	-20.6685
20	7	30	4	10	1.53	11.2	-3.69383	-20.9844
4	8	20	6	5	0.73	7.3	2.73354	-17.2665
26	9	30	8	10	2.21	12.4	-6.88785	-21.8684
7	10	20	8	5	0.9	8.1	0.91515	-18.1697
21	11	30	4	15	1.82	13.4	-5.20143	-22.5421
2	12	20	4	10	0.94	7.9	0.53744	-17.9525
11	13	25	4	10	1.32	10.1	-2.41148	-20.0864
14	14	25	6	10	1.54	10.3	-3.75041	-20.2567
5	15	20	6	10	1.1	8.6	-0.82785	-18.69
19	16	30	4	5	1.27	10.2	-2.07607	-20.172
24	17	30	6	15	2.3	12.6	-7.23456	-22.0074
27	18	30	8	15	3.02	14.8	-9.60014	-23.4052
10	19	25	4	5	0.83	8.9	1.61844	-18.9878
6	20	20	6	15	1.43	10.2	-3.10672	-20.172
15	21	25	6	15	1.63	12.1	-4.24375	-21.6557
9	22	20	8	15	1.74	11.1	-4.81098	-20.9065
18	23	25	8	15	1.97	12.9	-5.88932	-22.2118
16	24	25	8	5	1.45	9.9	-3.22736	-19.9127
25	25	30	8	5	1.92	11.5	-5.66602	-21.214
3	26	20	4	15	1.2	9.4	-1.58362	-19.4626
13	27	25	6	5	1.22	9.5	-1.7272	-19.5545

V. CONCLUSION

The Finite Element Simulation of resistance welding of HDPE considering the effect of three parameters such as current, pressure and welding time was done successfully. The following conclusions have been drawn from the results.

- As the value of current and time increases, heat penetration increases through the material. In each case, maximum temperature is noted at the centre area of mesh designed.

- As heat input increased, increase in temperature and accordingly increase in heat affected width are noted.
- Correspondingly structural analysis has been carried out and simulated values of displacement and strain has been found out.
- Temperature profile for each level of process parameters has been visualized, structural variations and different values obtained is analysed considering the number of experiments done.
- From the surface plots it is clear that the displacement and temperature is maximum for the higher value of the input parameters.
- From ANOVA,it can be found current is the most affecting in metal loss, bead height and bead width, and then after time is affecting and pressure is least affecting.

REFERENCE

[1] **Meng Hou, Mingbo Yang, Andrew Beehag, Yiu-Wing Mai and Lin Ye (1999) Resistance** welding of carbon fibre reinforced thermoplastic composite using alternative heating element. **Composite Structures 47, 667-72.**

[2] **higang Hou, Ill-Soo Kim, Yuanxun Wang, Chunzhi Li, Chuanyao** Chen Finite element analysis for the mechanical features of resistance spot welding process **Journal of Materials Processing Technology 185 (2007) 160–165**

[3] **anrong Kong, Radovan Kovacevic** 3D finite element modelling of the thermally induced residual stress in the hybrid laser/arc welding of lap joint. **Journal of Materials Processing Technology 210 (2010) 941–950**

[4] **Stavrov and H.E.N. Bersee (2005)** Resistance welding of thermoplastic composites-an overview**. Composites: Part A 36 , 39–54 K. Leskovics, M.Kollar, P. Barczy (2006)**

[5] **vijay Soundararajan, Srdja Zekovic, Radovan Kovacevic** Thermo-mechanical model with adaptive boundary conditions for friction stir welding of Al 6061 **International Journal for Machine Tools and manufacture 45 (2005) 1577- 1587.**

[6] **.K. Zhu, and Y.J. Chao** Effects of temperature-dependent material properties on welding simulation**, Computers and Structures, vol 80, 2002,pp 967–976.**

[7] **L.E. Lindgren,** 'Numerical modelling of welding'**, Comput. Methods Appl.Mech. Engrg., vol 195, 2006, pp 6710–6736.**

[8] **C.M. Chen, R. Kovacevic** Finite element modelling of friction stir welding- thermal and thermo mechanical analysis **International Journal for Machine and Manufacture 43 (2003) 1319 – 1326.**

[9] **C. Ageorges and L. Ye (2001)** Welding of composite/thermoplastic composite joints. **Composites: Part A 32, 1603-1602.**

[10] **M. Dube, P. Hubert, A. Yousefpour and J. Denault (2009)** Fatigue failure characterisation of resistance-welded thermoplastic composites skin/stringer joints. **International Journal of Fatigue 31, 719–725.**

[11] **P.J. Bates, S. Tan, G. Zak and M. McLeod (2009)** Shear strength and meltdown behaviour of reinforced polypropylene assemblies made by resistance welding. **Composites: Part A 40, 28–35**

[12] **Christophe Ageorges, Lin Ye, Yiu-Wing Mai and Meng Hou (1998)** Characteristics of resistance welding of lap-shear coupons. Part I. Heat transfer. **PII:s1359-835X(98)**

[13] **Rnjbar Nodeh, S. Seajzadeh, A.H. Kokabi** Simulation of Welding residual stresses in resistance spot welding, FE modeling and X-ray verification. **Journal of material processing technology 205 (2008) 60-69**