Investigations on Joining of High Density Poly Ethylene Sheets using Resistance Welding Technique

K. Pradeep¹, Sudhy S Panicker², K Panneerselvam* and P. Asokan¹

¹Department of Production Engineering, National Institute of Technology, Tiruchirappalli 620015, Tamil Nadu, India

²Department of Mechanical Engineering, Adi Shankara Institute of Engineering and Technology, Kalady 683574, Kerala, India

*kps@nitt.edu

Abstract. As a result of growing potential for high performance applications, parts made of HDPE materials is gaining greater interest for industry. Hence, the studies on joining of HDPE has got much significance. In this work, resistance welding of HDPE sheets was performed focusing on the governing set of parameters, current, pressure and time. Design of experiment (DOE) is done using full factorial design. The Analysis of Variance (ANOVA) was done with shear strength of weld as main parameter with the assistance of MINITAB software. Optimum parameters were identified in the study were medium level parameters produced good strength and better finished welded samples. Microstructure taken through Scanning electron microscopy (SEM) was supportive for this observation as there was a uniform flow of HDPE material without any void around the implant SS mesh for the best weld condition. The Finite element (FE) modeling of the resistance welding process was also performed using ANSYS to predict the thermal distributions occurring during the process depending on different levels of current and time inputs.

Keywords: HDPE, resistance welding, ANOVA, FE modeling

1. Introduction

High-density polyethylene (HDPE) is a thermoplastic prepared from petroleum products. HDPE is a commonly used thermoplastic material which has a wide variety of application in several industries. HDPE has become a leading polymeric material for numerous components due to several beneficial properties above metals such as light-weight, very good chemical and corrosion resistance, ease of joining, wide availability, higher shelf life, shorter processing times, and excellent reusability [1–6]. Gas and water distribution pipes, chemical-resistant piping systems, coax cable inner insulators (dielectric insulating spacer), geothermal heat transfer piping systems, etc. are some applications of HDPE products. Also, HDPE material is used in manufacturing several medical-related appliances. As this material has these many environment friendly applications, the studying of joining HDPE material structures is essential [7].

Numerous methods of adhesive bonding, mechanical fastening and fusion bonding are applied for joining the HDPE structures. The major issues like stress localisation and increase in weight during
mechanical fastening may be avoided by using adhesive bonding technique. However, adhesive bonding requires good pre-joining surface preparation and produces ambiguous joining quality. Moreover, adhesive bonding is also very sensitive to the chemically inert thermoplastics [8]. Fusion bonding is a method where the interface temperature at the joint crosses a definite value, i.e. \( T_g = \) glass transition temperature for amorphous polymers or \( T_m = \) melting temperature for semi-crystalline polymers. Further details of different conventional and non-conventional methods adopted to perform fusion welding of thermoplastics could be found in Jafrey and Panneerselvam [7,9]. Among these methods, resistance welding, also called as electrical-resistance fusion or electro-fusion or resistive implant welding, is an interesting technique for welding thermoplastics [1,3,10].

In resistance welding, an electrically resistive implant is placed between the bonding surfaces of the base material to generate the desired heat input at the joint [1,11,12]. The resistive implant, when electric current is passed across it, generates heat energy \( E \) according to Joule’s Law, \( E = I^2 R t \), where \( R \) is the resistance of the implant, \( I \) = current and \( t \) = elapsed time. Resistance welding is a rather simple technique and involves simple tooling and minimal surface preparation. When the energy supplied exceeds a threshold, the laminates temperature near to bonding surfaces starts to rise and with increase in welding time or current, the heat penetrates into inner layers of the material. It is advantageous to keep this heat affected zone (HAZ) closer to the bonding surface to avoid probable fibre disruption in the laminates. Resistance welding is usually applied for joining comparatively large and complex parts that does not require close-loop weld joints. This technique is effectively applied in the aerospace industry, especially by Fokker Special Products, the Netherlands [10,13]. Hence, the understanding of thermal bonding or welding behaviours of HDPE through resistance welding method is of high significance [1,2,8]. Recently, this technique was used by Xiong et al. [14] for hybrid joining of titanium alloy/thermoplastic composites using a stainless steel (SS) mesh implant. It was seen that rarely any study discussed a parametric feasibility on resistance welding of HDPE sheets under different levels of current, pressure and time used for joining.

Finite element analysis (FEA) is usually used in novel product design conceptualisation, and refinement of existing production line. FEA has become a solution to the task of predicting the behaviour of materials under various loading conditions. FE methods gives flexibility to designers by highlighting problem areas in the material and to understand the basis of deformations occurring within the material. Hence, it enables accurate prediction of structural deformations and temperature distributions happening within the investigated sample which are usually difficult to measure during actual physical experimentations [3]. Hence, this method aids in superior manufacturing technologies and to reduce the costs accrued during actual building and testing of samples. It was found that no previous work reported the investigations on FE simulation of resistance welding of HDPE sheets.

In this work, joining of HDPE sheets were performed through resistance welding process giving different levels of important input parameters, i.e. current, pressure and time. The analysis of variance (ANOVA) was done with the aid of MINITAB software. The full factorial combination approach was done to generate the experimental set of data. Mechanical characterization through tensile test, and metallurgical characterization using scanning electron microscopy (SEM) to assess better weld conditions. Finally, FE simulation was performed using ANSYS software to understand the temperature distribution occurring within the weld region.

2. Materials and methods

2.1. Specimen preparation

The HDPE sheets of dimension 150mm x 20mm x 4mm are produced using a semi-automatic injection molding machine shown in Fig. 1. The HDPE granules used for this purpose were Relene polyethylene grade M60075 from Reliance polymers. The standard properties of this polymer material were, melt flow 5t8 gm/10 minute, tensile strength 18 MPa, melting point 130-138°C, and continuous use temperature 180°C. Prefixed quantity of pure HDPE granules was fed to the heater and the heater
temperature was adjusted to 200°C so that easy flow of molten material is possible. Separate levers were provided to control the opening and closing of die and plunger movement.

2.2. Resistance welding setup

The schematic of the resistance welding arrangement is shown in Fig. 2a [1]. Actual dimensions of HDPE sheets and the heating element mesh used in the present study is shown in Fig. 2b. It consists of following standard components: HDPE sheets with sandwiched heating element, insulating slabs, tool for applying pressure, electric power supply and wires, clamping device, ampere meter and voltmeter. Fabric type stainless steel (SS) fabric type mesh of 120µm wire diameter with 20mm width was placed as the implant material. Ageorges et al. [11] had shown that better uniform heating in the weld interface was given by fabric type mesh than unidirectional ones. Stainless steel material was selected in this study as it resists rusting and will not get oxidised even at higher temperatures [8]. TORNADO201 electric welding machine was used, and the temperature generated in the mesh was calibrated with varying currents. Temperature generated in the mesh at different time intervals for 20A, 30A, 40A, 50A currents was found out using IR pyrometer as enlisted in Table 1. The maximum temperature reached for maximum current in minimum time condition.

![Figure 1](image1.png)

**Figure 1** Semi-automatic injection molding machine used to manufacture HDPE sheets

![Figure 2](image2.png)

**Figure 2** Schematic of (a) resistance welding setup [1] and (b) HDPE sheets specimen arrangement (all dimensions are in mm, representative figure and not to scale)
A pressure gauge was used to accurately measure pressure applied during the process. A fixture designed for perfect holding of HDPE plates and mesh implant are made having the dimensions 90mm X 90mm. Another main function of fixture is to transfer the pressure uniformly to the welding area. It has machined groves of 3mm depth and 20mm width, in which the plates are perfectly seated so that during weld time, it is hold tight. The mesh was placed between the sheet specimens as shown in Fig. 2b. This assembly was then placed into the fabrication set-up and tightened well after properly checking the alignment of plates. The current was switched off when nominal melting was achieved, and the joint was cooled under adequate pressure. The pressure applied during the welding enabled proper contact between the laminate surfaces at the bond line and promoted molecular diffusion at the joint. After the welding, the heating element was trapped inside the joint. The critical factors considered for the present study are current (I (A)), time (T (s)) and pressure (P (kg/cm²)).

2.3. Design of experiments
The full factorial approach was followed in the present study to perform number of experiments. This approach also facilitates the study of interactions between the factors. The present experiment was planned with has three factors, i.e. current, time and pressure varying in three levels as provided in Table 2. So, the total number of experiments conducted with different combinations was 27. The Design of experiments (DOE) was done with the support of MINITAB software. After making all arrangements, the current level was set to any one of the desired value. Then welding machine was switched on, measured the time to get it welded by means of a stop watch and switched it off after keeping it till the time reached as per DOE. With the welding set up and DOE, all 27 experiments are conducted, and all the welded specimens are shown in Fig. 3a. During the experiments, it was identified that the time required for melting is lower as melting starts with 20A current itself. With higher pressure, melting and joining occurs in 5s interval, and flames were formed at 12s. For very high values of current, time and pressure, liquid state HDPE spills out from the joint section as the melting occurs in a rapid rate.

### Table 1 Maximum temperature reached in different current values

| Current (A) | Maximum temperature (°C) | Time taken to reach max. temperature (s) |
|------------|---------------------------|----------------------------------------|
| 20         | 214                       | 100                                    |
| 30         | 260                       | 90                                     |
| 40         | 324                       | 85                                     |
| 50         | 354                       | 75                                     |

2.4. Finite element simulation
In this work, FE simulation was done using the widely used ANSYS software. The basic material properties assumed for FE modelling is given in Table 3. An important problem was the lack of material data available at high temperature. At first, values of the physical parameters of HDPE changing with varying temperature (0 – 700 K) were defined. Then the interpolation method as adopted to obtain the required thermal property parameters in the unknown temperature range. The melting point of HDPE material was assumed as 470 K. Please note, the temperature mentioned for FE simulations are in Kelvin scale. The welding is a non-linear transient state process, where the physical and thermal properties of
Figure 3 (a) HDPE specimens after welding and (b) Macrographs of specimens after tensile tests

Table 3 Properties of HDPE at different temperatures

| Sl No. | Temperature (K) | Thermal conductivity (W/mK) | Density (kg/m$^3$) | Young’s modulus (MPa) |
|--------|----------------|----------------------------|-------------------|----------------------|
| 1      | 300            | .460                       | 965               | 1400                 |
| 2      | 400            | .470                       | 955               | 1200                 |
| 3      | 425            | .480                       | 950               | 1150                 |
| 4      | 475            | .490                       | 950               | 1100                 |
| 5      | 500            | .500                       | 945               | 900                  |
| 6      | 550            | .510                       | 940               | 800                  |
| 7      | 600            | .515                       | 935               | 700                  |
| 8      | 675            | .520                       | 935               | 650                  |

the welding material rapidly changes with temperature. The controlling equation of heat conduction is given in Eq. 1.

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( \lambda \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \lambda \frac{\partial T}{\partial z} \right) + Q$$

(1)

where, $\rho$, $c$ and $\lambda$ are density, specific thermal capacity and heat conductance of material respectively. These terms were defined as functions of temperature and $Q$ was the power of heating element. The solid 70 (Brick 8 node 70) element having 3-D thermal conduction definition was adopted for the present FE modelling. The element also compensated for mass transport during heat flow from a constant velocity field. The heat generation rates were given as element body loads at the specified nodes.

The FE model was created as three distinct parts as shown in Fig. 4a. Two models of dimensions 150mm x 20mm x 4mm form the two work pieces to be welded together. The part in the middle portion which represents the SS mesh implant having dimensions 20mm x 20mm x 0.2mm are arranged as in experiment. Further, meshing was done such that the middle element is fine meshed and the other parts are coarse meshed, as represented in Figs. 4b,c. A free mesh, without any restriction of element shapes, and unspecified pattern was used with automatic meshing tool with a fineness level of 6/10. Heat generated in the heating element was calculated and given as input to this particular part. At the initial condition, i.e. $t = 0$, the work piece was given uniform initial temperature, which was the room temperature $= 300K$. Then, the convection heat transfer was assumed on the surfaces with surrounding temperature as 300K and convective heat transfer coefficient assumed was 25 W/m$^2$K. Ideal conditions
Figure 4 (a) FE model of resistance welding parts, (b), (c) view of the parts after meshing

Table 4 Calculated heat generation according to the current input

| Current (I) | Heat generated (kJ) |
|------------|---------------------|
| 20 A       | 642600              |
| 25 A       | 803250              |
| 30 A       | 963900              |

were selected at all levels of simulation process, where the parameters current time and pressure were given as inputs (all in SI units). As mentioned earlier, the middle part designed for mesh is acting as the heat source since heat generated in that area only which will melt the two welding parts and joining occurs. Heat generation occurring because of the current and voltage applied through stainless steel mesh calculated as follows. The SS mesh of 120microns with specified contained 120 threads. Hence, the heat generated depended only on 120 threads. As the welding machine supplied a constant output voltage of 63 V with an efficiency of 85%, the total power produced by mesh was calculated using Eq. 2.

\[
P = \frac{V \times I \times 0.85 \times 120}{0.2 \times 10^{-3}}
\]  

(2)

As the welding was a transient based analysis, another important parameter was time. This parameter was directly fed into the programme as sub-step time. The heat generated for three levels of current as given in Table 4 was applied to the defined intervals.

3. Results and discussion

3.1. Shear strength testing

To verify whether the welding had achieved improvement and to determine how good the approximation matched actual performance, all 27 samples were subjected to tensile loading. This is one of the most important and generally used method to measure the ability of a material to resist breaking/shearing under tensile load. Shear strength testing of all 27 specimens were done using UTM D60 with digital load-displacement data collection facility. Table 5 enlists tensile test data such as maximum strength, maximum elongation, and maximum load conditions for all the 27 welded samples. The macrograph of the test specimen before and after the test is shown in Fig 3b. The maximum strength and elongation of each specimen was recorded, and it was found that the shear strength of the welded samples varied from 18 MPa to 24 MPa confirming that weld strength is either greater than or equal to the tensile strength specified for the base material. Fig. 5a and b shows load-displacement plot for specimen showing high and moderate strength. However, the failure happened at heat affected zone (HAZ), proving the material strength less at this region. The coupon failure mode [8,15] was observed for fractured samples, as observed in Fig. 3b with the SS mesh intact at the joint. The working parameters, i.e. current, pressure, time corresponding to these welds were 30A, 6kg/cm², 10s, and 25A, 8kg/cm² and 10s, respectively. Visual inspection of the welded samples revealed that specimens welded with medium level parameters
corresponded to strength of 20 to 22 MPa showed better appearance and finishing. Breakage load was found to be the ultimate load itself, and fractured specimens showed brittle nature.

3.2. Evaluation of optimum welding parameters

The analysis of variance (ANOVA) was done using MINITAB software with factors affecting strength (current, pressure, time) as inputs. The purpose of ANOVA was to determine the factors and combination of factors that significantly affect the welding process. The regression equation obtained is shown in Eq. 3, with $R^2$ value of 75.86%. It should be noted that the linear model was adopted for simplicity of the problem.

$$\text{Strength} = 14.1 + 0.056 \text{Current} + 0.559 \text{Pressure} + 0.056 \text{Time}$$  \hspace{1cm} (3)

| Sl. No. | Current (A) | Pressure (kg/cm$^2$) | Time (s) | Strength (MPa) | Max elongation (mm) | Max Load (kN) |
|---------|-------------|-----------------------|---------|----------------|---------------------|---------------|
| 1       | 20          | 4                     | 5       | 2.125          | 0.5                 | 0.17          |
| 2       | 20          | 4                     | 10      | 20.625         | 8.7                 | 1.65          |
| 3       | 20          | 4                     | 15      | 20.625         | 4.72                | 1.65          |
| 4       | 20          | 6                     | 5       | 22.125         | 8.12                | 1.77          |
| 5       | 20          | 6                     | 10      | 22.875         | 8.6                 | 1.83          |
| 6       | 20          | 6                     | 15      | 20.625         | 6.92                | 1.65          |
| 7       | 20          | 8                     | 5       | 16.125         | 4                   | 1.29          |
| 8       | 20          | 8                     | 10      | 22.500         | 6.78                | 1.8           |
| 9       | 20          | 8                     | 15      | 18.750         | 6                   | 1.5           |
| 10      | 25          | 4                     | 5       | 19.125         | 4.9                 | 1.53          |
| 11      | 25          | 4                     | 10      | 23.250         | 8.88                | 1.86          |
| 12      | 25          | 4                     | 15      | 17.625         | 4.4                 | 1.41          |
| 13      | 25          | 6                     | 5       | 21.000         | 7.16                | 1.68          |
| 14      | 25          | 6                     | 10      | 21.750         | 19.2                | 1.74          |
| 15      | 25          | 6                     | 15      | 19.875         | 5.96                | 1.59          |
| 16      | 25          | 8                     | 5       | 21.000         | 38.8                | 1.68          |
| 17      | 25          | 8                     | 10      | 21.125         | 0.5                 | 1.69          |
| 18      | 25          | 8                     | 15      | 21.375         | 8.2                 | 1.71          |
| 19      | 30          | 4                     | 5       | 19.125         | 5.48                | 1.53          |
| 20      | 30          | 4                     | 10      | 18.000         | 6                   | 1.44          |
| 21      | 30          | 4                     | 15      | 18.000         | 4.69                | 1.44          |
| 22      | 30          | 6                     | 5       | 21.375         | 5                   | 1.71          |
| 23      | 30          | 6                     | 10      | 24.000         | 5.52                | 1.92          |
| 24      | 30          | 6                     | 15      | 13.125         | 8.64                | 1.05          |
| 25      | 30          | 8                     | 5       | 21.000         | 6.8                 | 1.68          |
| 26      | 30          | 8                     | 10      | 18.750         | 4.59                | 1.5           |
| 27      | 30          | 8                     | 15      | 18.000         | 3.61                | 1.44          |
Figure 5 Representative load-displacement plot of welded specimens at (a) medium strength and (b) high strength conditions

Table 6 Analysis of Variance for Strength

| Source           | DF | SS   | MS   | F      | P    |
|------------------|----|------|------|--------|------|
| Current          | 2  | 23.43| 11.72| 0.86   | 0.460|
| Pressure         | 2  | 47.00| 23.50| 1.72   | 0.239|
| Time             | 2  | 56.90| 28.45| 2.08   | 0.187|
| Current*Time     | 4  | 104.13| 26.03| 1.91   | 0.203|
| Current*Pressure | 4  | 41.65| 10.41| 0.76   | 0.578|
| Pressure*Time    | 4  | 70.02| 17.51| 1.28   | 0.353|
| Error            | 8  | 109.22| 13.65|        |      |
| Total            | 26 | 452.35|      |        |      |

$S = 3.69491$ \hspace{1cm} R-Sq = 75.86\% \hspace{1cm} R-Sq(adj) = 21.53\%$

Figure 6 Plots of (a) residual, (b) mean effects and (c) interaction for strength
From the generated values using MINITAB, ANOVA for strength is as given in the Table 6. Residual plot and mean effect plot for strength generated are as also shown in Fig. 6a and b, respectively. Among the parameters, current and time were found to be more significant. From the main effects plot it was seen that pressure is not as significant as current and time as the slope gradient is comparatively less. The interaction plot (Fig. 6c) also showed that better strength was obtained for the average values of current (=25A) and time (=10s) and pressure (=8kg/cm$^2$). Similar conclusions was drawn from the surface plots of strength with influencing factors, i.e. strength vs time, current (Fig. 7a), strength vs pressure current (Fig. 7b), and strength vs time, pressure (Fig. 7c). Corresponding surface response plots are shown in Figs. 7d-e.

Contour plot of strength vs current, time (Fig 7d) shows strength is better in the value of current ranging from 24A to 30A, and time range of weld in 8s to 15s. Fig. 7e shows that strength is maximum in pressure range 5.2 kg/cm$^2$ to 7.9 kg/cm$^2$ and for current values between 22A to 28A. The contour plot of strength vs pressure, time in Fig. 7f implied that strength is high for pressure varying from 5 kg/cm$^2$ to 8 kg/cm$^2$ and the welding time ranging from 9s to 13s. Thus, by the analysis of parameters used for the resistance welding experiments, optimized process parameters obtained for experiment number 17 where current = 25A, pressure = 8 kg/cm$^2$ and time = 10s for the specimen that produced shear strength of 21.125 MPa (Table 5).

3.3. Microstructure investigation

Microstructures at the cross section of welds were taken using scanning electron microscopy (SEM). The lap region were cut using diamond cutting wheel and the regular polishing procedure was performed for SEM analysis [7,8]. SEM images of a poor joint which are having lower level parameters (current=20A, pressure=4kg/cm$^2$, time=5s), medium level parameters (current=25A, pressure=6kg/cm$^2$, time=10s) and high level parameters (current=30A, pressure=8kg/cm$^2$, time=15s) are presented in Fig. 8a, b, and c respectively. The defective joint shown in Fig. 8a shows large amount of voids at the interface, i.e. between the inter layers of SS mesh and the parent metal. The flow of melted base material through the heated mesh is not uniform and the joining was not intact between two parts of specimen. So, the sample showed poor strength properties. Similar observation was also reported by previous researchers [7,8,15] as well. However, the implant material (SS mesh) was completely embedded without any voids in the case of medium and high level parameters. A proper flow of molten base material through the heated mesh is not uniform and the joining was not intact between two parts of specimen. The welding with medium level parameters maintained the heat affected zone (HAZ) closer to the bonding surface that avoided probable fibre disruption in the laminates.
3.4. Temperature distribution

The FE modelling was used to predict the temperature generation during the present resistance welding process. A clear visualisation of the heat distribution and movement of heat flux occurred inside the welded material and clear visualisation was obtained. Nodal temperature distribution was the transferring of heat from the heating element through the material. Temperature values were plotted at the middle part of the welded specimen. Maximum temperature was visualised at a little distance away from the centre. This was because of the spacing of nodes generated by the program and heat generated is applied across the node. The temperature values are listed against the depth of the material and part

Figure 8 Microstructure at joint of resistance welded samples of (a) low level, (b) medium level, and (c) high level parameters.

Figure 9 (a) FE simulated representation of temperature profile developed in resistance welded sample, (b) corresponding temperature distribution across the weld and (c) surface plot of Temperature v/s Current, Time
of the material above melting point. Fig. 9a shows the FE simulated image of temperature profile developed within the welded sample for the optimum welding parameter condition (i.e. for experiment 17, refer to section 3.2). It is clearly visible in Fig. 9b, that the temperature distribution was uniform across the depth at the middle section of the specimen. Similarly, FE simulations for three levels of current (20A, 25A, 30A) and for three levels of time (5s, 10s, 15s) were performed. The maximum temperature recorded for these conditions are tabulated and plotted as shown in Fig. 9c. Maximum temperature was obtained for the parameters 30A, 15s condition. It was observed that the maximum temperature obtained from simulation was in the similar range of that obtained for mesh temperature at different time intervals measured with the help of infrared pyrometer (Table 1). The differences in values occurred because of several assumptions used for FE simulations and actual values may differ according to the process conditions. The consideration of vaporization of the material at elevated temperatures were also negated while FE modeling.

4. Conclusions and future scopes
In this work, the effect of current, pressure and time on the resistance welding properties of HDPE sheets are studied. The FE simulation of HDPE resistance welding considering welding current and time was performed to predict the temperature distribution. Following are the major conclusions drawn from the study.

- Experimental setup was designed to perform lap joint resistance welding of HDPE sheets with SS mesh as implant material. DOE was done using full factorial approach, and ANOVA was carried out to obtain optimum welding parameters of current, pressure and time corresponding to shear strength data obtained through tensile testing.

- From different plots such as residual, main effect, interaction, surface and contour plots, it was concluded that the optimum resistance welding condition was for experiment number 17, where current = 25A, pressure = 8 kg/cm² and time = 10s for the specimen that produced shear strength of 21.125 MPa. The failed samples showed coupon failure and the fracture was brittle in nature.

- The SEM images revealed that there was a uniform flow of HDPE material without any void at the weld region showing good adhesion of both the plates with perfect implanting of SS mesh. The visual inspection and SEM images were also supportive to the ANOVA results showing that resistance welding with medium level parameters produced better finished components with reliable strength. This was because the heat affected zone (HAZ) was maintained closer to the bonding surface that avoided probable fibre disruption within the laminates.

- The temperature profile obtained for each level of process parameters were visualized through FE simulations. The heat penetration increased with increase in the value of current and time. The central area of mesh showed maximum temperature and these predicted temperature were in almost matching with the experimentally measured mesh temperature under corresponding current and time values.

Performing a coupled thermal-structural analysis is important future work for quantitative prediction of temperature-dependent variation of size and shape of the HDPE sheets. In this context, FE simulations also considering melt flow and vaporization of the HDPE material at elevated temperatures will give more accurate predictions of resistance welding process. Moreover, a future work on preparation, optimization of the parameters, mechanical and microstructural characterization, and corresponding FE analysis of resistance welding of HDPE nanocomposites is an important aspect considering the potential of applied polymer composites.

References
[1] Stavrov D and Bersee H E N 2005 Compos. Part A Appl. Sci. Manuf. 36 39–54
[2] Kumar S, Kumar Y and Singh A 2014 Int. J. Eng. Tech. Res. 2 89–96
[3] Ageorges C and Ye L 2001 J. Thermoplast. Compos. Mater. 14 449–475
[4] Offringa A.R 1996 Compos. Part A Appl. Sci. Manuf. 27 329–336
[5] Zehtabayazdi A, Zebardj S M, Sajjadi S A and Abolfazli Esfahani J 2007 Express Polym. Lett.
192–97

[6] Chang I Y and Lees J K 1988 J. Thermoplast. Compos. Mater. 1 277–296
[7] Jafrey D D and Panneerselvam K 2018 J. Adhes. Sci. Technol. 32 2350–2363
[8] Panneerselvam K, Aravindan S and Noorul Haq A 2012 Mater. Des. 41 453–459
[9] Jafrey D D and Panneerselvam K 2017 Polym. Eng. Sci. 57 495–504
[10] Nino G F, Ahmed T J, Bersee H E N, Beukers A 2009 Compos. Part B Eng. 40 237–248
[11] Ageorges C, Ye L and Hou M 2000 Compos. Sci. Technol. 60 1191–1202
[12] Villegas I F, Moser L, Yousefpour A, Mitschang P and Bersee H E N 2013 J. Thermoplast. Compos. Mater. 26 1007–1024
[13] Mathijsen D 2016 Reinf. Plast. 60 405–407
[14] Xiong X, Zhao P, Ren R, Zhang Z, Cui X and Ji S 2019 Diam. Relat. Mater. 101 107611
[15] Hou M, Yang M, Beehag A, Mai Y and Ye L 2006 Compos. Struct. 47 667–672