Numerical and Experimental Study on the Influence of Low Frequency Vibration on Strains of Single Point Incremental Forming

Riham Ali Nema, Mauwafak Ali Tawfik and Muthanna Hamzah Sadoon
Mechanical Engineering Department, University of Technology, Baghdad, Iraq
E-mail: 20040@uotechnology.edu.iq

Abstract. Single point incremental forming (SPIF) is a decent potential procedure appropriate for small batch production. In addition to that, the formability of the material is expanded compared to the other sheet metal forming processes. Meanwhile, the more advanced SPIF process requires perfect comprehension of the material deformation technique. In this paper, the equipment of vibration - sheet metal forming was designed. Finite element model was used to demonstrate and analyze the effect of different frequencies of vibration on the strain distribution in single point incremental forming process. The results show that specific vibration parameters can decrease strain. Moreover, an experimental vibration system using single point incremental forming was designed. The numerical model is primarily supported by test results conclusions from a simple cone of Al1050 aluminum alloy.

Keywords. Numerical, Frequency vibration, Single point, SPIF, Al1050 aluminum alloy.

1. Introduction
Recently, industry has a developing interest for sheet metal forming process depicted by a high flexibility which grants diminishing creation times and lower production costs particularly while dealing with a little batch size, redone parts and prototypes. Single Point Incremental Forming, SPIF, has an extraordinary prospect in mechanical enforcements because of its straightforwardness [1]. If an incremental plastic forming process system is utilized, the vibration sheet metal opposition will be extremely reduced, a production quality amended; and sheet metal incremental forming procedure range will be grown up. Then, a new approach for the high rigidity and strength sheet metal forming will be set [2]. Finite element method and procedure reaction such as thickness distribution, geometrical accuracy, and surface roughness were investigated the incremental forming process [3]. An investigation was carried out to determine the implementation of the finite element (FE) method to foretell thickness and stress distribution on the sheet through forming to reinforce the information about the ISF approach [4]. A fulfillment survey of superfine division of a conical shape was achieved by the implementation of the incremental forming process and the finite element method was used for the sake of validation to estimate the geometry limits of the final product [5]. The data and major outcomes of research on the influence of utilizing pre-forming blank in SPIF via FEA were obtained. The cited SPIF had been calculated by
definite circumstances pertaining to the test work piece, tool, etc., applying ANSYS11 [6]. Overall finite element model has been accomplished to analyze the condition of the strain and stress in the nearness of the contact region, where the plastic deformation extends by means of the forming tool action [7]. In this work, vibration was excreted to the sheet metal so as to alter the persistent extrusion liaison between the tool head and the aluminum plate (AL1050). Range of vibration frequencies on strain distribution were investigated by applying numerical simulation and experiments.

2. Experimental work equipment
Tests were performed using a 3-axis CNC vertical milling machine. The sheet metal was clamped by specific designed steel equipment to avoid material flow into the forming area (Fig.1).

Figure 1. 3D CNC milling machine.

All the test information was carried out to produce a simple geometry cone. Material, AL1050 with 1mm thickness was used to obtain a cone shape shown in figure.2 (a, b) whose dimensions are rotating base of 100 mm in diameter, a height of 30 mm and draw angle of 45°. The sheet metal with (300x400) mm is clamped into a fixture, with supporting plate including a 105mm diameter orifice to allow the formed cone inside it. A rectangular edge was used to clamp the sheet. The sheet was not allowed to move towards the forming region, while the available material within the 105mm diameter can be influenced by the forming tool. The tool rotates at a speed of 200 rpm and moves along a path that keeps track of the cone contour at a feed rate of 400 mm/min. A slammed vertical step size of 0.25 mm and a tool diameter of 12 mm were implemented. Elsewhere, the parameters might be varied and applied. The vibration system used in this work consists of Out-of-balance semi disc which is mounted to DC motor rigidly attached to the plate. Speed control unit was applied to adjust the excited frequency on the plate with frequency range of (20, 40, 60, 80,100) Hz.
3. Numerical simulation

Referring to the finite element method, each element is depicted by complex differential equation describing its application approximated by algebraic expressions which link the forces to the displacements for each element, and then construct into matrices for the whole system.

\[ \{F\} = [K] \{D\} \]

Where; \( \{F\} \) is the force vector matrix, \( \{D\} \) - is the displacement vector matrix, and \( [K] \) - is the stiffness matrix.

Initially, stiff and formable based regions on the blank and specified tool geometry must be restricted. Then the geometry of the formable part is meshed into finite element. The incremental sheet metal forming process is simulated using ANSYS 18.0 with a general-purpose of FE package. The complexity of tool and sheet contact, friction between them, kind of material and the nonlinearities of the geometry were taken into account through the problem of modulation. An axisymmetric model was utilized for the purpose of computational and size reduction of the model using ANSYS program application to generate the tooling geometry. The implemented element type to represent the work piece was (plane182). Element target 169 and contact 172 were utilized to represent the complex interaction between the sheet and the tool. The center line of blank sheet was kept in x-direction, while the fixture was held in both x and y-directions. Then, according to CAD profile the incremental displacement of the tool was applied. Coulomb friction is depicted for the friction model, and the performed tensile test was used to define the true stress-strain curve via ANSYS input (Sinusoidal vibration) which acts on the plate at range of frequencies. A pure Aluminum (AA1050) is utilized in the present work. The mechanical properties of the blank are presented in table (1).

| Blank material properties used in FE model. |
|--------------------------------------------|
| Density (\( \rho \)) | Young’s modulus (\( E \)) | Poisson’s ratio (\( v \)) | Yield stress (\( \sigma_y \)) | Tangent modules (\( E_t \)) |
|------------------------|-----------------|----------------|-----------------|-----------------|
| 2700 kg/m^3            | 75 GPa          | 0.33           | 78 MPa          | 0.2 GPa         |
4. Result and discussion

4.1. Effect of vibration on strain

The distribution of effective strains is studied in the formed part. It was used to analyze the numerical model of toughness over SPIF. The commencement of vibration made the formation technique of SPIF very complicated. The continual squeezing contact in the metal sheet and the tool head was changed with the aid of alternation contact due to the sinusoidal wave. Lubricant liquid is used to reduce the effect of friction between the tool and the sheet. As a result of that, vibrancy reasons widespread enhancements of lattice dislocation density and the dislocation ratio via grain refinement leading to improvement in the deformation of sheet in imitation to yield better formability. Figure (3) illustrates the finite element model by using ANSYS software which shows the strain distribution without vibration.

![Figure 3. Strain distribution for numerical part obtained by ANSYS software without vibration.](image)

The vibrations (sin wave) was applied at the center of the plate. The strain distribution for simulations with vibration for different values of frequencies are presented in figures (4, 5, 6, 7, 8) respectively. When vibrations were applied, the numerical results showed the expected reduction of strain at the implemented range of frequency. With vibration frequency (20Hz) the maximum von mises strain increased from 0.47594 to 0.4921, and the increase of the value of strain was 3.3%. When the vibration frequency was increased from 20 Hz to 40Hz, the strain decreased to 0.4573, i.e., it was reduced by 3.9% from the strain without vibration. As the frequency (60Hz) was applied, the strain decreased to 0.4682 with 1.63% from the value of strain without vibration. When the vibration frequency increased to 80 Hz, the value of maximum strain reduced from 0.47594 to 0.44748, namely it was reduced at 5.98%. The strain increased again to 0.49307 at the frequency of (100Hz), at 3.47% from the strain without vibration. From the cited figures, it is noticed that using a frequency of 80 Hz leads to reducing the strain of the forming shape more than the other frequencies at the cited range. From this point of view, it is obvious that the frequency range is very important to improve the strain distribution in the formed shape.
4.2. Numerical and experimental results validation

In order to examine the strain in the final product, the primary blanks have been gridded into (4x4) mm mesh. A quarter from the final forming shape have been cut by wire cut machine to simplify the measured thickness of the shape by measuring the gridded after forming. The strain in three directions can be calculated. At last the von misses effective strain has been calculated according to the below equation.
\[ \varepsilon_e = \frac{\sqrt{2}}{3} \sqrt{(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2 + 6\varepsilon_{xy}^2} \] (2)

Where; \( \varepsilon_x, \varepsilon_y \) and \( \varepsilon_z \) are strains in x, y and z directions respectively, and \( \varepsilon_{xy} \)-strain in xy direction.

Figure 9 (a, b, c, d and e) shows a comparison between the numerical and experimental results. The average variance between them for frequencies (20Hz, 40Hz, 60Hz, 80Hz and 100Hz) are (10.3%, 8.95%, 9.3%, 9.36%, 9.8%) respectively. Hence, the agreement between them is acceptable. When the applied frequency is equal to (80Hz), the magnitudes of strains indicate less values of strain distribution for both the numerical and experimental results.

**Figure 9.** Experimental and numerical strains versus the mesh regions at range of frequencies.
5. Conclusions

1- Vibration has great influence on the strain distribution of the deformed shape produced by single point in incremental forming process.

2- It was found that the reduction in strain values of the deformed shape at frequency of 80 Hz was (5.98%) compared to that without vibrations. This percentage is the lowest among the implemented range of frequencies.

3- The experimental strain distribution results are acceptable in comparison with the numerical results after applying the vibration.

6. References

[1] J Naranjo et al 2015 Analysis and simulation of Single Point Incremental Forming by ANSYS (Published by Elsevier Ltd, Procedia Engineering) vol 132 pp 1104–1111

[2] Gaipin Cai, Ningyuan Zhu and Na Wen 2011 Stress Analysis of Sheet Metal Vibration Incremental Forming (Advanced Materials Research) vol 154–155 pp 166–170

[3] Cai Gaipin, Xing Congwen, Jiang Zhihong and Zhang Zhongkai 2012 Sheet Single Point Vibration Incremental Forming Process Simulation And Analysis Of Process Parameters (Advanced Materials Research) vol 430–432 pp 74–78

[4] Salah B M Echrif and meftah Hrairi 2011 Process Simulation and Quality Evaluation of Incremental Sheet Forming (IIUM Engineering Journal, Special Issue, Mechanical Engineering)

[5] Qasim Mohamed Doss, Tahseen Fadhel Abaas and Aqeel Sabree Bedan 2013 An Investigation Study of Thinning Distribution in Single Point Incremental Forming Using FEM Analysis (Al-Khwarizmi Engineering Journal) vol 9 no 3 pp 1–14

[6] K I Abass 2016 A Study On Using Pre-Forming Blank In Single Point Incremental Forming Process By Finite Element Analysis (20th Innovative Manufacturing Engineering and Energy Conference (IManEE 2016) IOP Conf. Series: Materials Science and Engineering 161, 012031, doi:10.1088/1757-899X/161/1/012031)

[7] D M Neto, J M P Martins, M C Oliveira, L F Menezes and J L Alves 2016 Evaluation of Strain and Stress States in the Single Point Incremental Forming Process (Int J Adv Manuf Technol), DOI 10.1007/s00170-015-7954-9