Al-Khwarizmi Engineering Journal, Vol. 9, No. 3, P.P. 1 -14 (2013)

Al-Khwarizmi Engineering Journal

An Investigation Study of Thinning Distribution in Single Point Incremental Forming Using FEM Analysis

Qasim Mohamed Doss* **Tahseen Fadhel Abaas****
Aqeel Sabree Bedan***
**College of Engineering / University of Baghdad*
,Department of Production Engineering and Metallurgy / University of Technology*

(Received 14 April 2013; accepted 29 May 2013)

Abstract

Single Point Incremental Forming (SPIF) is a forming technique of sheet material based on layered manufacturing principles. The sheet part is locally deformed through horizontal slices. The moving locus of forming tool (called as toolpath) in these slices constructed to the finished part was performed by the CNC technology. The toolpath was created directly from CAD model of final product. The forming tool is a Ball-end forming tool, which was moved along the toolpath while the edges of sheet material were clamped rigidly on fixture.

This paper presented an investigation study of thinning distribution of a conical shapes carried out by incremental forming and the validation of finite element method to evaluate the limits of the process as regards to the geometry of the final product.

Three conical products have been carried out during this study with different forming angles and depth. The process was simulated using FEM program (ANSYS 11.0) and the results showed that the deviations between simulated and real values did not exceed of 6%.

Keywords: *Single Point Incremental Forming (SPIF), Experiments, Thinning, Finite element method*.

1. Introduction

Single point incremental forming (SPIF) process is interesting both industrially and scientifically. In the first case, sheet metal components can be manufactured without specific tools using a CNC milling machine. This kind of process can be produced complex parts in small batch or for single part. In particular, SPIF used as a rapid manufacturing process to custom-made parts. However, this advantage is limited by the important thinning of the sheet, the occurrence of defects. [1]

In the SPIF process, thickness of formed part is evaluated by Formula (1) which is called Sinlow formula. This is usually utilized for the shear forming process.

$$t_f = t_o \sin(\frac{p}{2} - y) \qquad \ldots(1)$$

Where, (t_f) is final thickness, (t_o) is initial thickness and (ψ) is wall angle.

The formula calculation shows that the thickness of vertical wall is zero. Therefore, the capacity of SPIF process only can deform the wall angle less than 90 degrees.

The parts having various steep walls are very difficult to apply this process. Because the thickness of deformed part is calculated uniformly by Formula (1) and the thickness of black is constant.

Significant advantages of this process over conventional forming include greater formability, low forming forces and generic tooling configuration. One of the major research problems of considerable interest to the sheet metal forming community is the accurate prediction of fracture in SPIF. This is important because an underestimation of the fracture depth will result in a loss of the advantage of enhanced formability of the process and an overestimation will cause component failure during the forming process itself. Furthermore, a better physical understanding of the mechanisms of deformation and fracture in SPIF is of great importance since this can aid the choice of appropriate process parameters for the process and can lead to modifications of the process to further enhance the achievable formability. Aspect Different steps of this process are shown in Figure (1). [2]

Numerical investigations using FEM were also conducted to investigate the final thickness and mechanisms in SPIF. This process modeled the contact between the tool and the sheet using a moving ball end tool method.

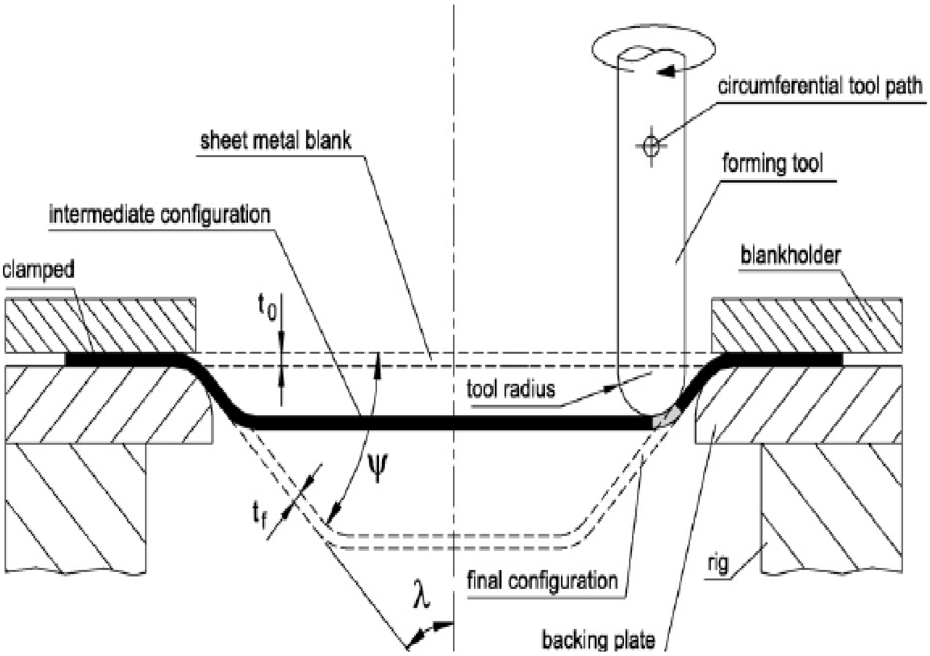


Fig. 1. Single Point Incremental Sheet Metal Forming Process.[2]

2. Material and Testing

2.1. Material

For this study, the selected material is an Aluminum (Al-1050) sheet with an initial thickness of (0.9mm) and an initial hardness of (24 HV). Its composition studied in State Company for Inspection and Engineering Rehabilitation activities (S.I.E.R)), and is given in Table (1).

Table 1,
Chemical Composition of Aluminum 1050 Sheet.

Material		Si%	Fe%	Cu%	Mn%	Mg%	Cr%	Ni%	Zn%	Al%
Al-1050	**Exp.**	0.142	0.315	0.013	0.013	0.001	0.001	0.003	0.006	99.5
	Iso	0-0.25	0-0.4	0-0.05	0-0.05	0-0.05	0-0.03	0-0.03	0-0.07	99.5

2.2. Tensile Test

In order to simulate the SPIF process by means Finite Element Program, the value of some parameters was measured in tensile test. In practical, the mechanical characterization was performed with tensile testing machine, such as tensile strength of (90MPa) and young modules of (72 GPa) have peen obtained. [3]

Tensile test has been down in the University of Technology- production engineering and metallurgy, strength of material Lab; Figure (2) presents tensile test setup. These tests result in (90^o to rolling direction) because the strength in this way is minimum according to literatures. [4] While, most of experimental work was done in isotropic condition due to Axisymmetric product.

Table (2) defines the result of tensile test for Aluminum sheet.

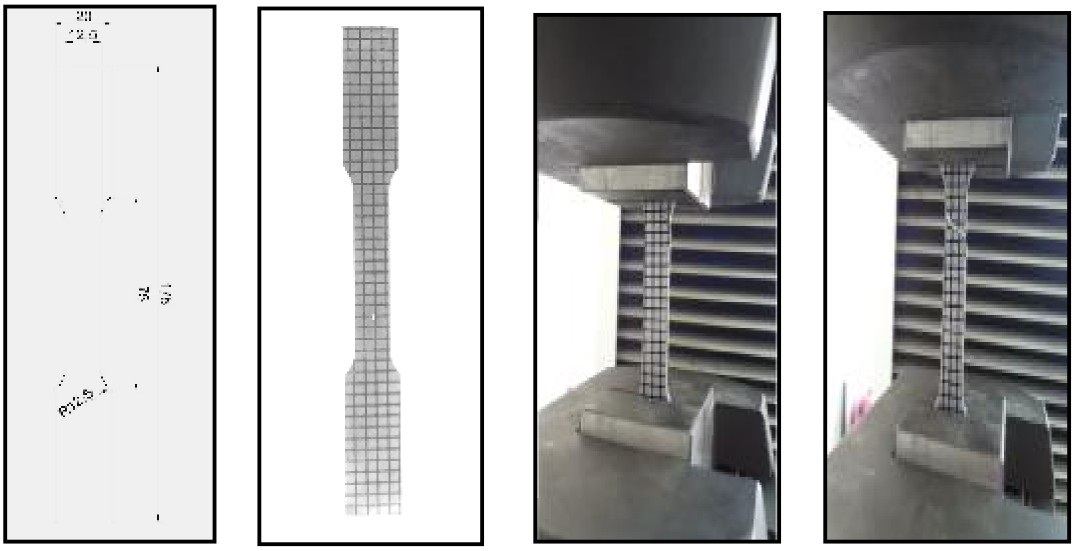


Fig. 2. Tensile Test of Al-1050.

Table 2,
Mechanical Properties for Al-1050 Sheet.

Material		Tensile Strength Mpa	Modulus of Elasticity Gpa	Poissons Ratio	Elongation % on 50 mm G.L.	Vickers Hardness VPN	Iso
Al-1050	**Exp.**	90	72	0.33	37	24	Al-1050-‘o’
	Iso	80-100	70-75	0.33	35-42	20-30	

2.3. Experimental Device and Forming Strategies

2.3.1. Experimental Device

To validate the numerical simulations, experiments were conducted. For this purpose, a dedicated apparatus was designed and realized. The representation of this tooling is done in Figure (3). It is composed of a fixed die support, a modular die, a fixed blank holder clamped with the die by screws and the forming tool (ball end tool).

The modular defined different shapes depending on the geometry of the part to be produced. In particular, it limited the non-desired bending obtained on the base of the part by specifying the nearest contour of the final part.

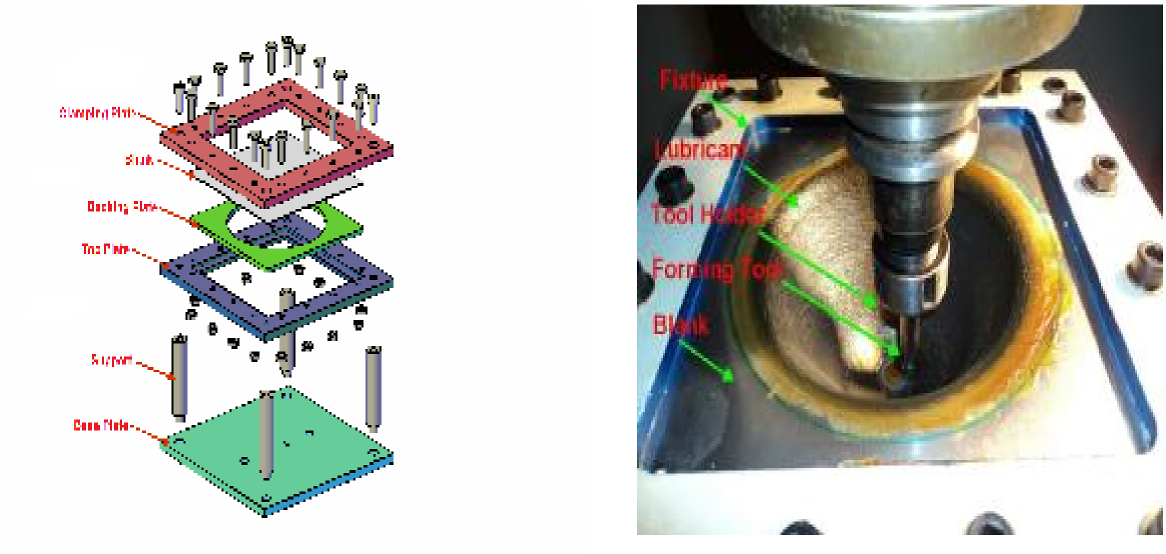


Fig. 3. Illustrated the Proposed Clamping Device.

2.3.2. The Shape of the Part

The carried out with a cone with different depth and slope angles, beginning from a square sheet with a side of 225 mm, the tool paths, whose example are illustrated in Figure (4) are characterized for forming of cone is a sequence of circular coils generates the tool path. The first of which presents (D=160mm) and feed a long depth has a step (0.3mm) and final circle is (10mm).

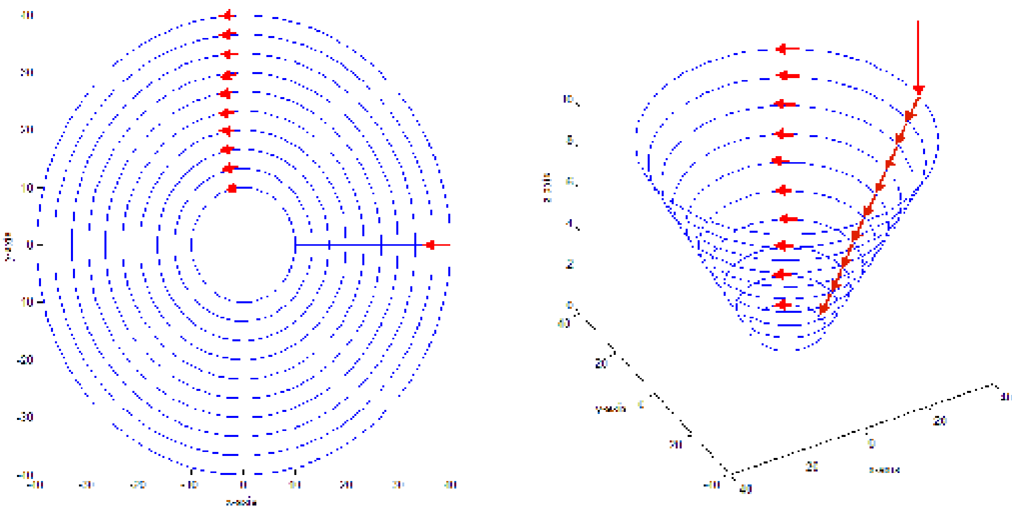


Fig. 4. Isoplaner Tool Path. [5]

A conical shape is proposed to investigate the single point incremental sheet forming of thin sheet. The geometry definition is illustrated in Figure (5 &6). This shape has been chosen because there is alternation between the (x, y) and the z directions during the forming process. Therefore it will be possible to determine the influence of the tool position on the forming thinning.

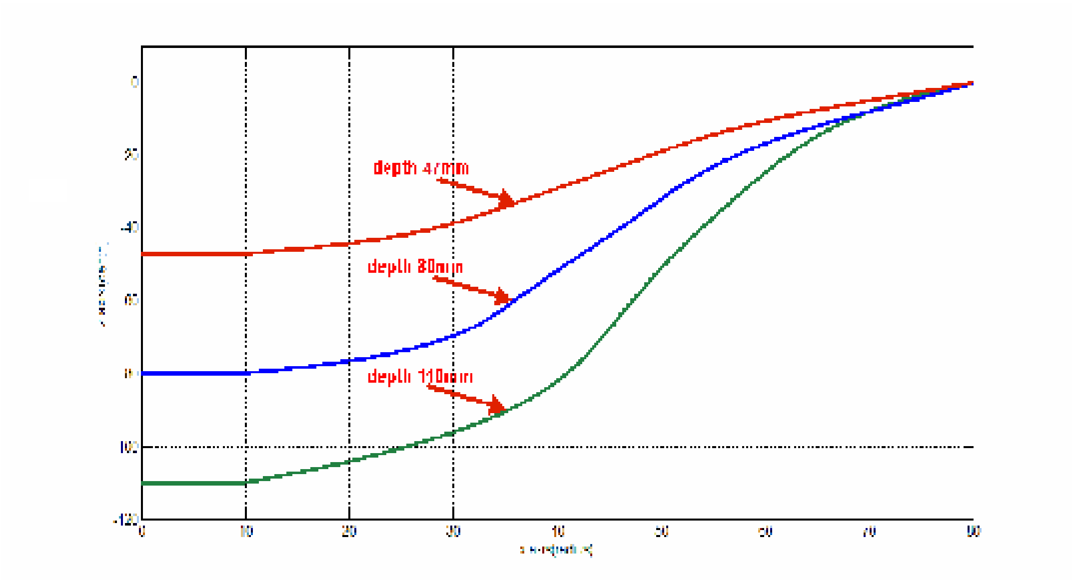


Fig. 5. Proposed Conical Cup Profile used in this Research (Depth =47, 80 &110mm).

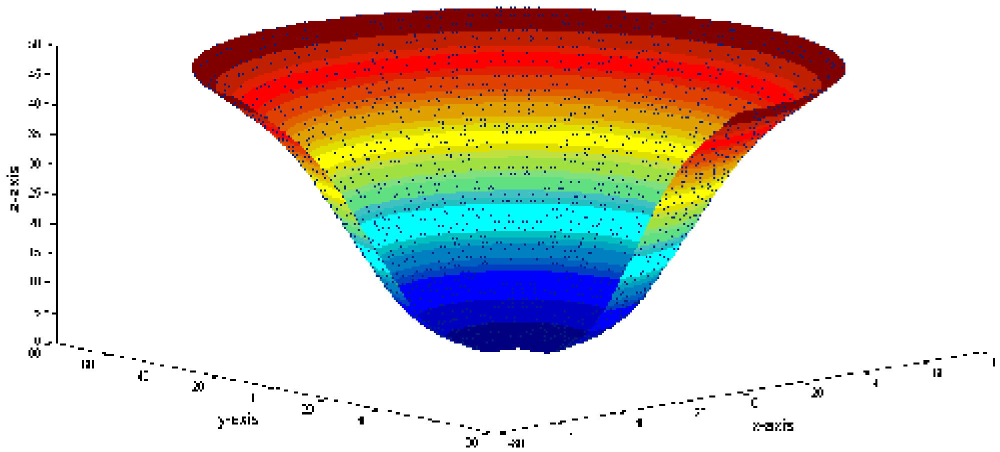


Fig. 6. Proposed 3D- CAD Model used in this Research (Depth=47mm).

2.3.3. Forming Strategies

The forming was carried out on vertical CNC machine using steel tool (Tool Steel Material(x210)) with Ball end diameter (12mm), Different strategies are possible to produce the part. For the conical shape, one approach was considered: Isoplaner toolpath.

Figure (7) shows a photograph of an experimental test, for every test the tool has been put in rotational speed of (100r.p.m) and a feed rate of (750mm/min) was set out.

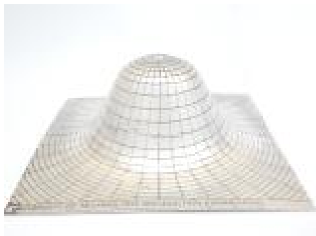
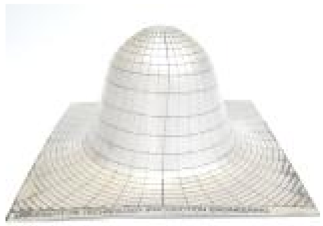
	Depth (47mm)	Depth (80mm)	Depth (110mm)
Isoplaner Toolpath			
	Actual forming time=55.5 min	Actual forming time=95.4 min	Actual forming time=134.9min
Maximum Forming angle	41.9^{o}	63^{o}	78^{o}

Fig. 7. Geometrical Parameters.

3. Numerical Methods

An overview of the analysis method used in the research is given in the chart in Figure (8). The chart presents in detail the phase of preprocessing the data corresponding to the physical model of the forming process. Complex differential Equations describing the applicable physics can be approximated with algebraic expressions within each element. In the case of structural finite element analysis (FEA), these simplified expressions relate forces to displacements within each element; the expressions are then assembled into matrix form,

$$F = [K] * D \qquad \ldots(2)$$

Where *(F)* is the force vector applied to a structure, *(D)* is the resultant displacement vector, and *[K]* is the stiffness matrix of the system.

At the beginning, there are defined the formable and stiff bodies based on the blank's and the tool geometry. The geometry is the one specific for the moment of process initiation. Based on the geometry of the formable body, this is meshed into finite elements. [6] To the set of elements thus defined, the following are also associated:

- Material data, specifically the tensile test.
- Geometrical data, namely the date of profile.
- Define boundary condition and type of element; based on recommendations from the specialty literature.

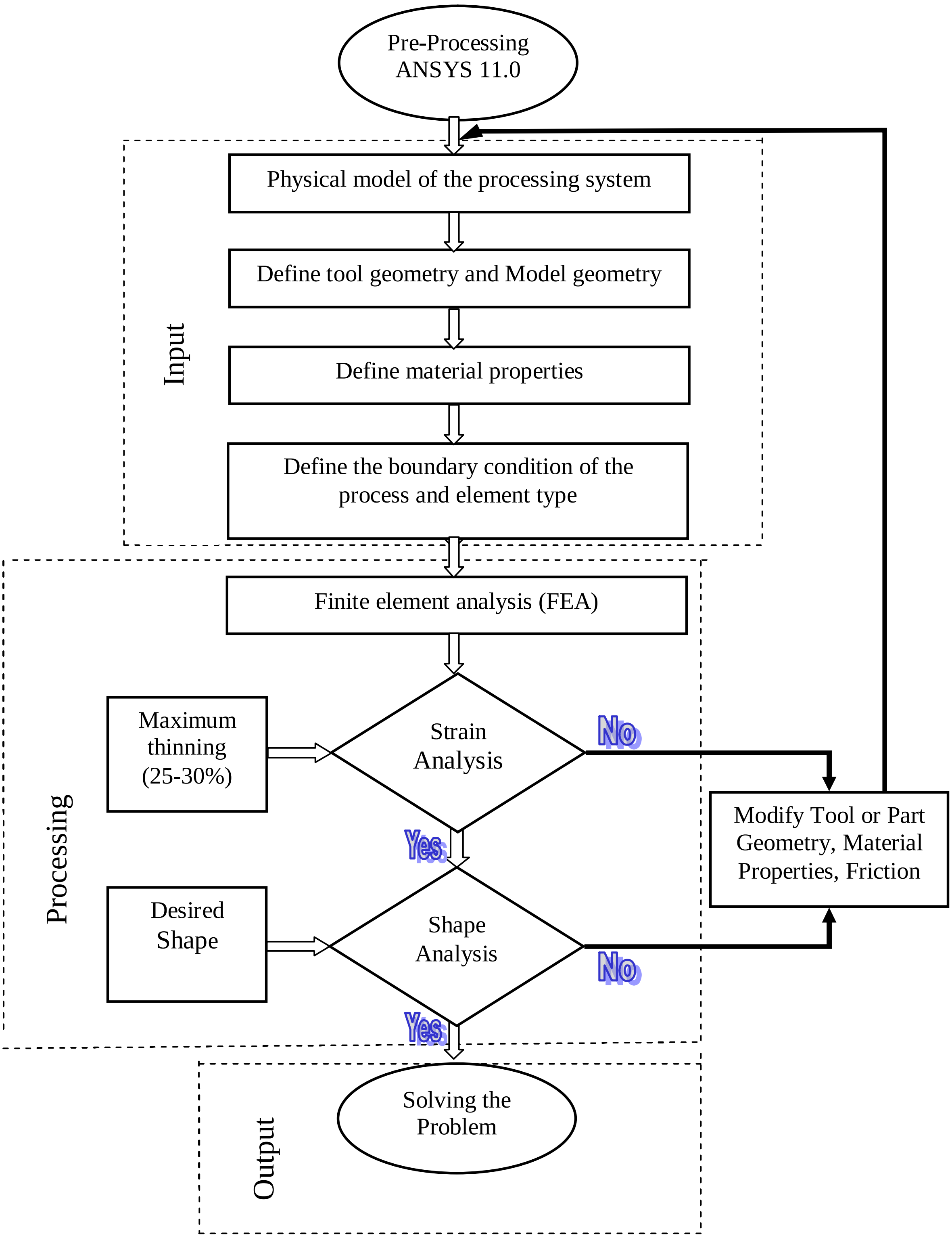


Fig. 8. Simulation Flowchart that Illustrated Input, Analysis, and Output Condition in Formability Analysis.

3.1. Defined Blank Element Type

The element used to simulate the blank is (VISCO106), VISCO 106 is used for 2-D modeling of solid structures, and it is defined by four nodes having up to three degrees of freedom at each node (X, Y and Z).

3.2. Material Properties

A pure Aluminum (AA1050) was used in this work, the specific mechanical properties result from Stress-Strain curve from tensile test illustrated in Table (3).

Table 3,
Show the Material Properties of the Blank Used in FE Model.

Density	ρ	2700 kg/m3
Young's modulus	E	75 GPa
Poisson's ratio	ν	0.33
Yield stress	σy	78 MPa
Tangent modules	Eτ	0.2 GPa

3.3. Boundary Conditions

3.3.1. Displacement and Loading

1. The die was held fixed by nodal constraints in the x and y-direction n1, n2 and n3.
2. The Fixture was constrained so as to allow movement in the x,y-direction.
3. The blank was hold by fixture at n4 and n5, and hold in center line and allow to movement in y-axis only at n6.
4. The punch motion was specified in curve profile with constant speed.

Figure (9) illustrated the FE model and Boundary condition.

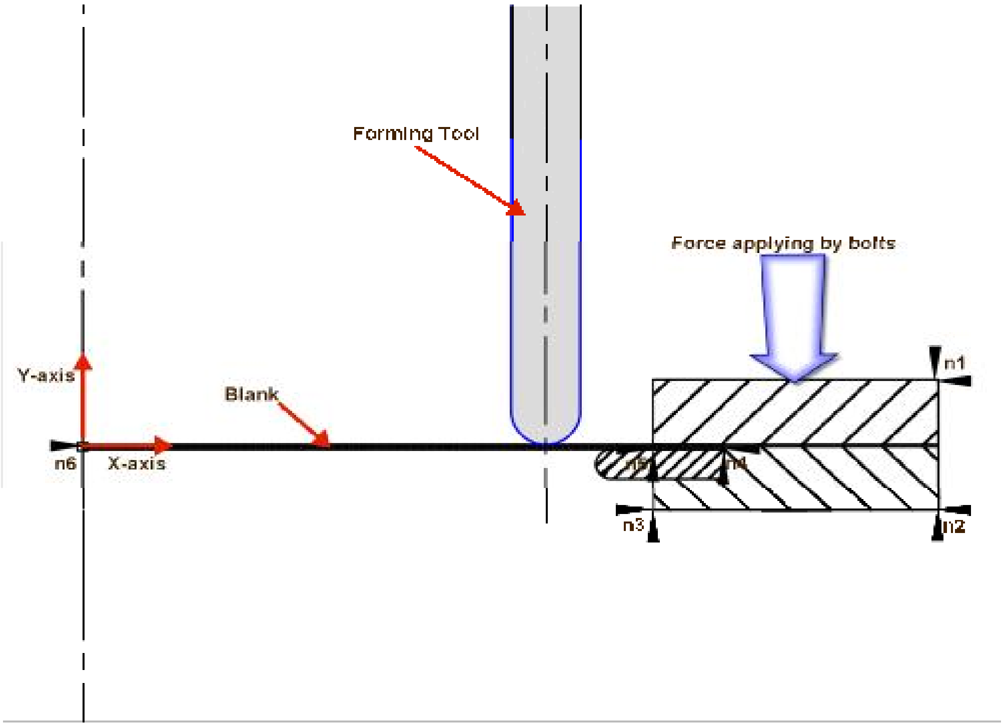


Fig. 9. Show Finite Element Boundary Conditions.

3.3.2. Coulomb Friction Model

In this work, the value of friction coefficient (0.04) was used for simulation model.

3.3.3. Contact

Three contact interfaced were defined between the tool, die and blank.

1. Contact between tool and upper blank surface.
2. Contact between tool and upper blank surface-lower blank surface die interface.
3. Contact between blank holder and upper blank surface –lower blank surface die interface.

The contact between the tool, blank holder, and die illustrated in Figure (10). The tool and work piece like rigid to flexible contact, and from ANSYS the nod-to-surface contact model and element target 69 and contact 71 are used to represent the contact between these reigns. Figure (11) present the FE-output while Figure (12) is a plot of a typical stress legend used to describe the motion of the tool to a depth of 110 mm. and the applied blank holder force using Screw.

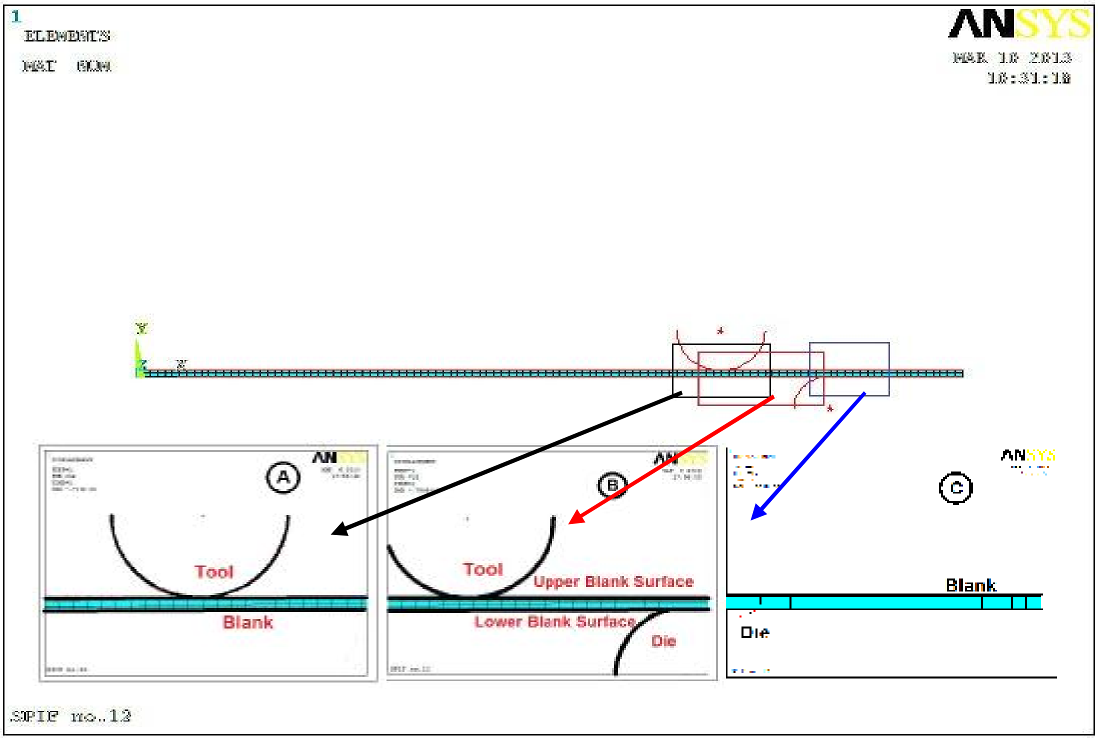


Fig.10. Illustrated Three Type of Contact used in Incremental Forming.

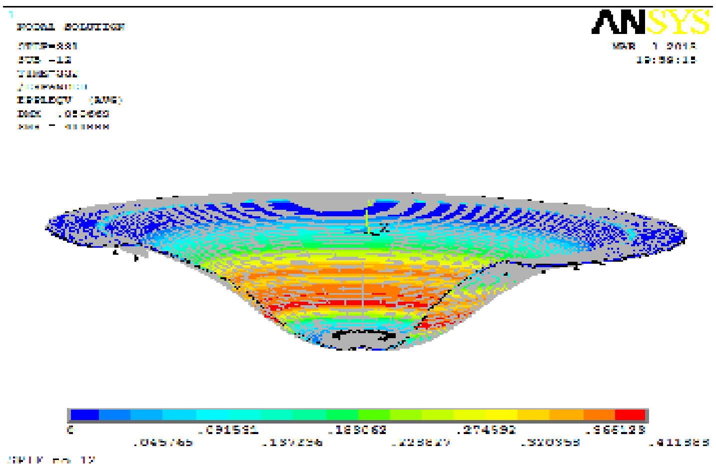


Fig. 11. Simulate Forming Part Output from ANSYS Package.

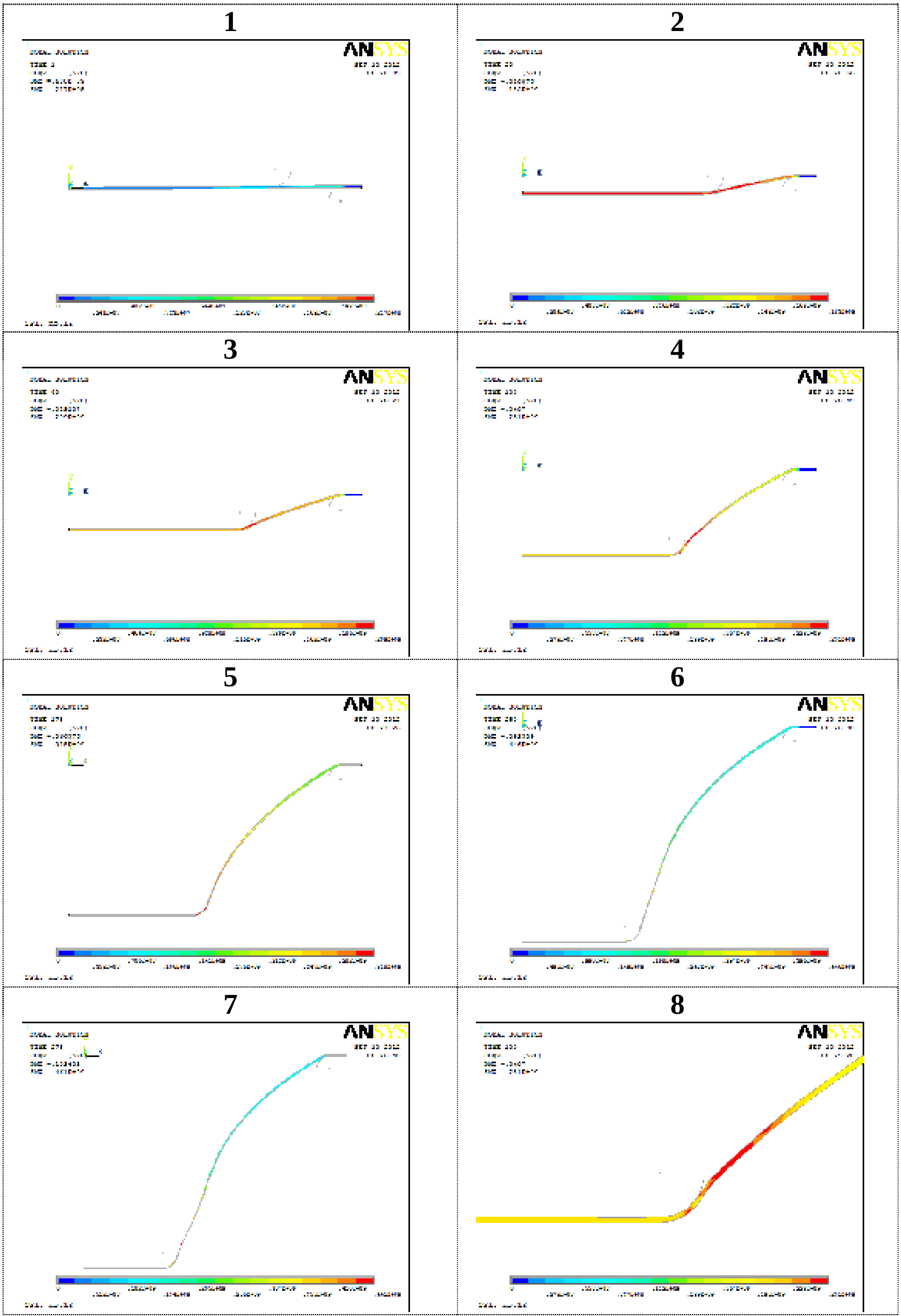


Fig. 12. Sequences of Incremental Forming Model using ANSYS 11.0.

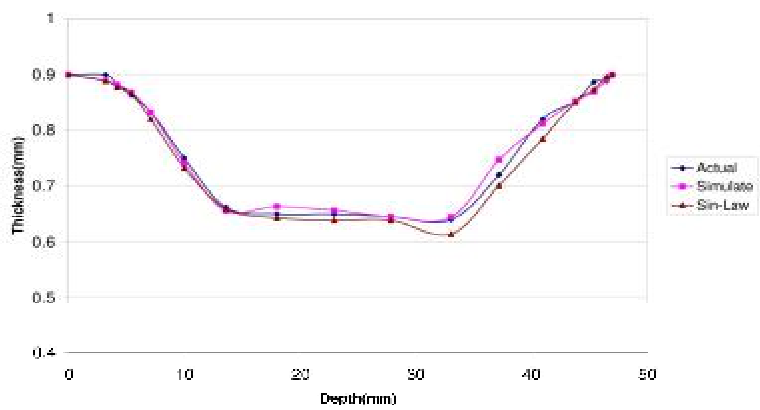


Fig.13. Illustrated Thickness Distribution for Forming Depth (47).

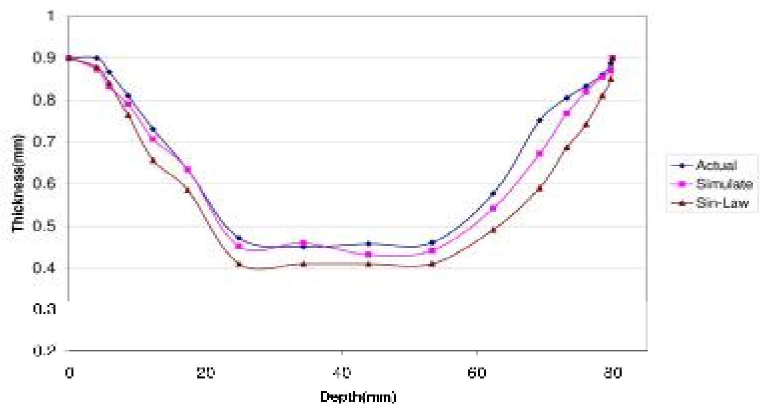


Fig. 14. Illustrated thickness distribution for forming depth (80)

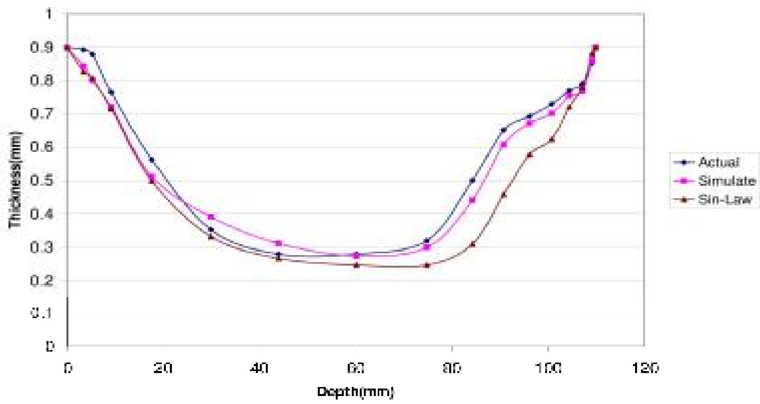


Fig. 15. Illustrated Thickness Distribution **for Forming Depth (110).**

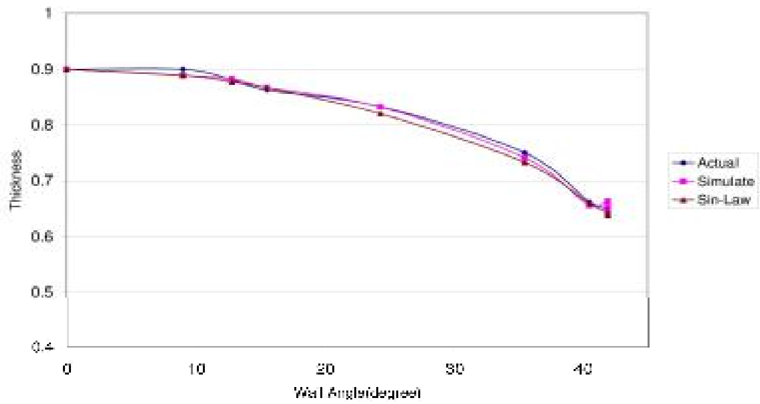


Fig. 16. Illustrated Thickness Distribution for Maximum Forming Angle (41.9o).

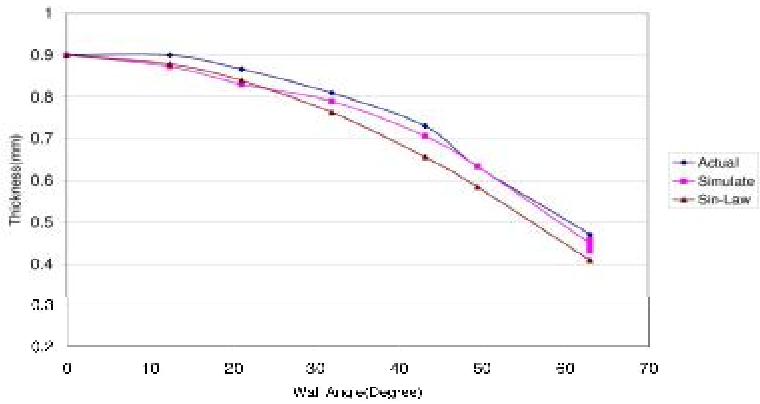


Fig. 17. Illustrated Thickness Distribution for Maximum Forming Angle (63o).

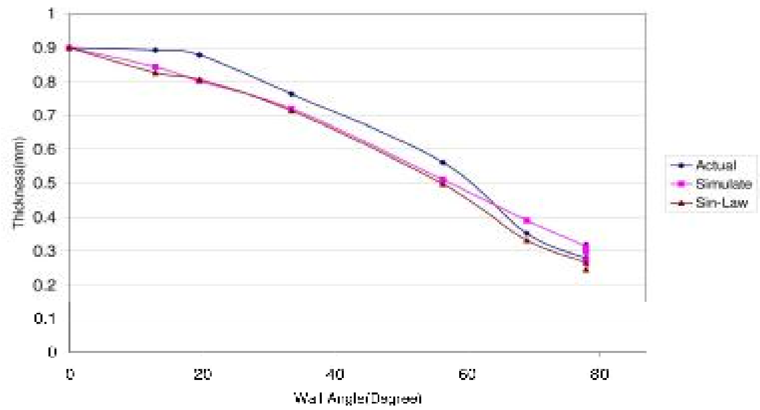


Fig. 18. Illustrated Thickness Distribution for Maximum Forming Angle (78o).

4. Results Conclusions

1. FEM was fully applied to the simulation of a cone whose results were compared to the theoretical (Sin Law) and to experimental measurements in terms of thickness.
2. From the results shown in Figures (13,14 and 15) it clearly seen that the predicted behavior of thinning distribution with depth are coincided well with both theoretical (Sin Law) and practical behaviors and the overall deviation did not exceed 6%.
3. Two types of analyses through the finite element method: analyses for determining the influence of depth parameter on thinning distribution, and analyses for determining the influence of wall angle on thinning distribution. That present when forming angle increase with respect to forming depth the thinning increase as shown in Figures (16,17 and 18).
4. The deviation between the actual thickness and simulate are increases with respect to depth of forming increase due to increasing of spring back and bending stress on internal wall of part.
5. Using FEM, the wall angle affected on thinning behavior, and the maximum thinning is locate in the maximum forming angle.
6. Using FEM, It was found that both through-the-thickness shear and local bending of the sheet around the tool plaied a role in fracture in the SPIF process, which happened at forming depth (120mm) when the product was failure.

5. References

[1] S. Thibaud, R. Ben Hmid, F. Richard and P. Malécot' "A fully parametric toolbox for the simulation of single point incremental sheet forming process: Numerical feasibility and experimental validation", ", Elsevier Ltd, Simulation Modelling Practice and Theory, Vol. 29, 2012.

[2] D. Xu, Rajiv Malhotra, N. Venkata Reddy, Jun Chen and Jian Cao, "Analytical prediction of stepped feature generation in multi-pass single point incremental forming", Elsevier Ltd, Journal of Manufacturing Processes, Vol. 14, 2012.

[3] Q. Mohamed Doss, Tahseen Fadhel Abaas and Aqeel Sabree Bedan, "The effect of tool path strategy on twist behavior in Single Point Incremental Sheet Metal Forming", Journal of Engineering, Baghdad University, Vol. 4, 2013.

[4] R. Malhotra, Liang Xue, Ted Belytschko and Jian Cao, "Mechanics of fracture in single point incremental forming", Elsevier Ltd, Journal of Materials Processing Technology, Vol. 212, 2012.

[5] C. Felipe Guzm, Jun Gu , Joost Duflou , Hans Vanhove , Paulo Flores and Anne Marie Habraken, "Study of the geometrical inaccuracy on a SPIF two-slope pyramid by finite element simulations", Elsevier Ltd, International Journal of Solids and Structures, Vol. 49, 2012.

[6] S. Dejardin, S. Thibaud, J.C. Gelin and G. Michel, "Experimental investigations and numerical analysis for improving knowledge of incremental sheet forming process for sheet metal parts", Elsevier Ltd, Journal of Materials Processing Technology, Vol. 210 363–36, 2010.

دراسة توزيع التخصر في عمليات التشكيل النقطي المتزايد باستخدام طريقة تحليل العناصر المحددة

قاسم محمد دوس* **تحسين فاضل عباس**** **عقيل صبري بدن*****
*كلية الهندسة / جامعة بغداد
،*قسم هندسة الانتاج والمعادن / الجامعة التكنولوجية

الخلاصة

تعتمد طريقة التشكيل النقطي التزايدي على مبدا التشكيل للطبقات الافقية المتوازية وصولا للتشكيل النهائي للمنتج وذلك من خلال حركة عدة التشكيل "التي غالبا ماتكون عدة صلدة ذات نهاية كروية مصنعة من الفولاذ المقاوم" على مسار يعرف مسبقا باحدى طرق التصميم المعان بالحاسوب وتستخدم هذه الطريقة لتشكيل الصفائح المعدنية باستخدام مكائن التحكم الرقمي (المبرمجة).

في هذا البحث تم دراسة توزيع السمك لشكل مخروطي تم انتاجه باستخدام عملية التشكيل النقطي المتزايد وتم محاكاة العملية باستخدام برنامج التحليل للعناصر المحددة لتنبوء بالتوزيع الحاصل للمنتج النهائي.تم انتاج ثلاث اشكال مخروطية بزوايا تشكيل واعماق مختلفة عمليا ومحاكاتها باستخدام برنامج تحليل العناصر المحددة (الانسز), وتم التنبو بالشكل الناتج وتوزيع السمك نسبة لعمق التشكيل وزاوية الميلان والانفعال وبانحراف للقيم لايتجاوز ٦% عن القييم العملية والنظرية المحسوبة.