Manufacturing Science and Education 2025

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 68, Issue Special II, Month July, 2025

VALIDATION OF FINITE ELEMENT MODEL FOR STRAIN ANALYSIS OF SINGLE POINT INCREMENTAL FORMING PROCESS

Mihai-Octavian POPP, Gabriela-Petruța POPP, Anca Lucia CHICEA, Ilie Octavian POPP, Fineas MORARIU

Abstract: *Motivated by the need to improve strain distribution prediction in single point incremental forming process, this study investigates the results obtained from a numerical model developed, in comparison with experimental results. The problem addressed is the limited experimental validation of finite element predictions, which reduces confidence in process simulations. Using ABAQUS, a detailed model with the help of finite element method was developed and applied to truncated cone parts formed by a KUKA industrial robot. Strain and thickness data were obtained from experimental investigations using a 3D optical measurement system and compared to numerical predictions. The results showed slight overestimations of the numerical model, particularly in minor strains, but overall deviations remained within acceptable limits. The model reliably predicts strain distributions, confirming its applicability for predicting strain distribution in single point incremental forming process and improving manufacturing outcomes.*
Keywords: *Single Point Incremental Forming, Finite Element Analysis, Numerical Modeling, Experimental Validation, Strain Distribution Prediction.*

1. INTRODUCTION

Single point incremental forming (SPIF) has been extensively studied because it offers notable benefits, including reduced costs and greater flexibility compared to other cold plastic deformation techniques [1]. Despite this, its industrial application remains limited, mainly due to disadvantages such as lengthy production times and modest improvements in the geometric precision of the final parts [2].

Another study explores the Finite Element (FE) modeling of the SPIF process, focusing on reducing computational time while maintaining accuracy [3]. Using ABAQUS, an incrementally formed truncated pyramid was analyzed and experimentally validated [4]. Mass scaling and time scaling techniques were applied, and an artificial neural network (ANN) optimized mesh size and mass scaling to minimize bottom pillowing error. The results demonstrate that computational efficiency can be significantly improved without compromising geometric accuracy, offering a reliable approach for modeling SPIF processes [5]. Recent advances in manufacturing emphasize the need to select the optimal process for each application [6].

This research aims to create a numerical model of the SPIF process using finite element analysis and to validate it through experimental tests by examining the strain in the deformed parts. To achieve this, straightforward components shaped as frustum cones were analyzed.

2. MATERIALS AND METHODOLOGY

In this study, two frustum cone parts, with parameters detailed in Table 1, were formed using the SPIF process. The sheet blanks employed were made of 0.8 mm thick DC01 steel, known for its high plasticity, making it well-suited for plastic deformation. Subsequently, a numerical model was developed to simulate the process, and the resulting strains were examined to validate the model. To strengthen the validation, the technological and geometric parameters were deliberately varied between the two cases.

Table 1

Process parameters used.

Case no.	Punch diameter [mm]	Vertical step [mm]	Wall angle [°]	Total depth [mm]
1	6	0.25	50	30
2	10	0.75	60	40

Figure 1 shows the two frustum cone parts after the incremental forming process.

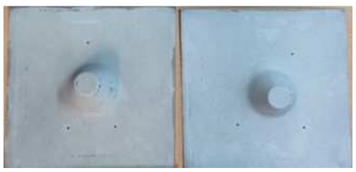

Fig. 1. Deformed parts through SPIF: case 1 left, case 2 right

To carry out the forming process, a KUKA KR 210 industrial robot with a rated load capacity of 2 kN was employed. The punch movement was programmed using SprutCAM software, generating a continuous spiral path for both sample cases. The experimental configuration included the robotic system, a custom-designed fixture to secure the sheet blanks in a vertical orientation, and a 3D ARAMIS optical system used to monitor strain evolution in real time.

For the numerical analysis, the explicit solver in ABAQUS software was utilized. The simulation model consisted of the sheet blank, the spherical punch, and the upper and lower retaining rings (see Fig. 2). To streamline calculations and meet research goals, the rings were treated as discrete rigid components, while the punch was modeled as an analytical rigid surface. Only the sheet blank was defined as deformable, with its mechanical properties derived from tensile tests performed on DC01 steel samples.

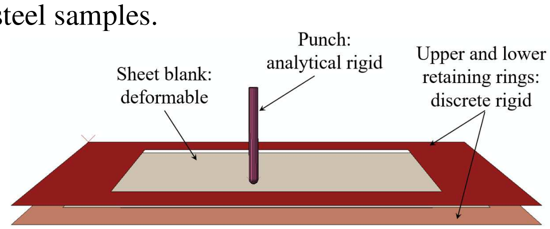

Fig. 2. Components modeled for the SPIF process simulation

3. EXPERIMENTAL AND SIMULATION DATA

Following the completion of both simulations and laboratory tests, several key indicators were analyzed, including major and minor strains, X and Y directional strains, shear angles, and material thinning. To assess the validity of the numerical model, the percentage differences between the predicted and measured values were computed using the following relation:

$$Deviation[\%] = 100 - \frac{simulation_{value} \cdot 100}{experimental_{value}} \quad (1)$$

Table 2 summarizes the maximum values recorded for each parameter in both scenarios, displaying side-by-side the experimental results, the numerical predictions, and the associated percentage differences.

Table 2

Comparison between experimental values, numerical predictions, and relative errors (maximum values reported for both cases).

Result Type	Case no.	Exp. value [%]	Numerical value [%]	Deviation [%]
Major strain	1	58,8	64,36	9,46
	2	113,7	124,13	9,17
Minor strain	1	5,22	5,9	13,01
	2	8,8	9,99	13,52
X strain	1	57,94	60,34	4,14
	2	107,4	112,25	4,52
Y strain	1	57,99	62,11	7,10
	2	113,3	121,72	7,43
Shear angle	1	23,8	24,44	2,69
	2	37,04	38,16	3,02
Thick. reduct.	1	39,4	42,42	7,66
	2	55,07	58,86	6,88

The table presents a comparison between experimental and numerical results for strain components and thickness reduction in the SPIF process. The numerical values generally overestimate the experimental values, with differences ranging from approximately 2.69% to 13.52%. The largest discrepancies are observed in minor strain values, with deviations exceeding 13%, while the shear angle shows the smallest differences, staying below 3.1%. The thickness reduction follows a similar trend, with numerical predictions slightly higher than experimental results. Overall, the numerical model provides a reasonably accurate approximation, though refinements may be necessary to improve predictions for minor strain components.

Figures 3 and 4 show the major strain distributions measured experimentally for both SPIF cases. These graphics reveal the deformation patterns, emphasizing zones of higher and lower strain. By examining them, variations in material response and overall strain spread across the parts become visible.

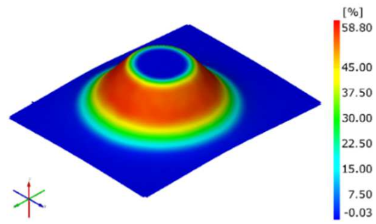

Fig. 3. Experimental distribution of major strain for case 1

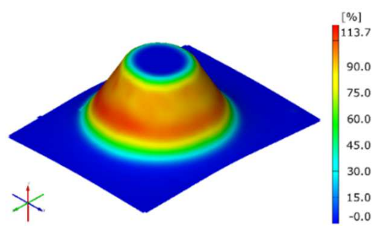

Fig. 4. Experimental distribution of major strain for case 2

Fig. 5 and 6 present the numerical predictions of major strain obtained using the Finite Element (FE) model developed in ABAQUS for both cases. These theoretical distributions allow for a direct comparison with the experimental results, helping to assess the accuracy of the numerical model. Differences between experimental and numerical results can indicate potential improvements in model calibration, such as refining material properties, boundary conditions, or mesh resolution.

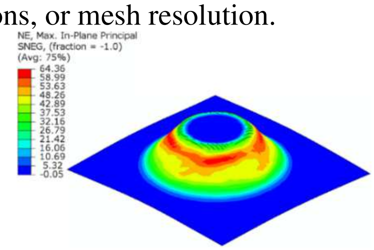

Fig. 5. Theoretical distribution of major strain for case 1

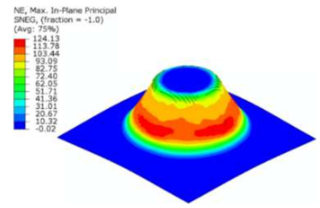

Fig. 6. Theoretical distribution of major strain for case 2

Fig. 7 and 8 show the numerical distribution of normal strains in the X direction, as predicted by the ABAQUS simulation for both cases. The normal strain component in the X direction is essential for understanding material flow and deformation characteristics along the forming direction. By comparing these numerical results with experimental data, the accuracy of the simulation in capturing strain evolution and deformation mechanics can be evaluated.

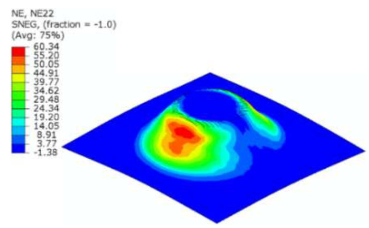

Fig. 7. Theoretical distribution of normal strains in the X direction for case 1

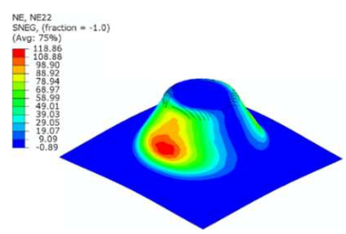

Fig. 8. Theoretical distribution of normal strains in the X direction for case 2

These graphical comparisons between experimental and numerical strain distributions play a crucial role in validating the Finite Element (FE) model and ensuring its predictive accuracy for the SPIF process. By overlaying or juxtaposing the experimental and theoretical strain maps, it is possible to assess how well the numerical model replicates real-world deformation behavior.

Another important factor in this comparison is the localization of strain concentrations, particularly in highly deformed zones. The ability of the numerical model to correctly predict these areas determines its reliability for further process optimization.

4. CONCLUSIONS

This study validated a finite element model capable of accurately predicting strain behavior in the SPIF process. While numerical predictions slightly overestimated experimental results, differences remained within acceptable limits, confirming the model's reliability. The work is important because it offers a practical tool for optimizing SPIF parameters and improving process efficiency.

Future research should focus on refining material models, improving mesh resolution, and testing more complex geometries to enhance predictive accuracy and broaden the model's industrial applicability.

5. REFERENCES

[1] Racz, S.-G., Breaz, R., Bologa, O., Tera, M., Oleksik, V. S., *Using an Adaptive Network-based Fuzzy Inference System to Estimate the Vertical Force in Single Point Incremental Forming*, International Journal of Computers Communications & Control, 63-77, 2019

[2] Pepelnjak, T., Sevšek, L., Lužanin, O., Milutinović, M., *Finite Element Simplifications and Simulation Reliability in Single Point Incremental Forming,* Materials, 1996-1944, 2022

[3] Bologa, O., Breaz, R., Racz, S.-G., *Using the Analytic Hierarchy Process (AHP) and fuzzy logic to evaluate the possibility of introducing single point incremental forming on industrial scale*, Procedia computer science, 408-416, 2018

[4] Crenganis, M., Breaz, R., Racz, G., Bologa, O., *Inverse Kinematics of a 7 DOF Manipulator Using Adaptive Neuro-Fuzzy Inference Systems*, Proceedings of the 2012 12TH INTERNATIONAL CONFERENCE ON CONTROL, AUTOMATION, ROBOTICS & VISION (ICARCV), 2012

[5] Bologa, O., Breaz, R., Racz, S.-G., Crenganis, M., *Decision-making tool for moving from 3-axes to 5-axes CNC machine-tool*, Procedia computer science, 2016

[6] Breaz, R., Bologa, O., Racz, S. -G., *Selecting between CNC milling, robot milling and DMLS processes using a combined AHP and fuzzy approach*, Procedia Computer Science, 2017

Validarea modelului teoretic a procesului de deformare incrementală într-un singur punct din punct de vedere ale deformațiilor specifice

Scopul acestei lucrări este de a îmbunătăți predicția distribuției deformațiilor în procesul de deformare incrementală într-un singur punct (SPIF), analizând rezultatele obținute dintr-un model numeric și comparându-le cu cele experimentale. Problema abordată este validarea experimentală limitată a predicțiilor realizate prin metoda elementelor finite, ceea ce reduce încrederea în simulările de proces. Utilizând ABAQUS, a fost dezvoltat un model detaliat cu ajutorul metodei elementelor finite și aplicat pe piese sub formă de trunchi de con, deformate cu un robot industrial KUKA. Datele privind deformațiile și subțierea materialului au fost măsurate cu ajutorul unui sistem optic de măsurare 3D și comparate cu predicțiile numerice. Rezultatele au arătat ușoare supraestimări ale modelului numeric, în special la deformațiile specifice secundare, dar abaterile au rămas în limitele acceptabile. Modelul prezice cu o bună precizie distribuția deformațiilor specifice, validând aplicabilitatea sa pentru predicția distribuției deformațiilor în procesul SPIF și pentru îmbunătățirea rezultatelor în cadrul procesului.

Mihai-Octavian POPP, Ph.D., Lecturer, Lucian Blaga University of Sibiu, Faculty of Engineering, Department of Industrial Machinery and Equipment, E-mail: mihai.popp@ulbsibiu.ro

Gabriela-Petruța POPP, Ph.D., Lecturer, Lucian Blaga University of Sibiu, Faculty of Engineering, Department of Industrial Machinery and Equipment, E-mail: gabriela.popp@ulbsibiu.ro

Anca Lucia CHICEA, Ph.D., Associate Professor, Lucian Blaga University of Sibiu, Faculty of Engineering, Department of Industrial Machinery and Equipment, E-mail: anca.chicea@ulbsibiu.ro

Ilie Octavian POPP, Ph.D., Associate Professor, Lucian Blaga University of Sibiu, Faculty of Engineering, Department of Industrial Machinery and Equipment, E-mail: ilie.popp@ulbsibiu.ro

Fineas MORARIU, PhD candidate, Teaching assistant, Lucian Blaga University of Sibiu, Faculty of Engineering, Department of Machines and Industrial Equipment, fineas.morariu@ulbsibiu.ro