DOI: 10.71762/jgjf-5b81

Research Paper

Finite Element Analysis and Experimental Factors Influencing Human Face Profile Shaping in Single-Point Gradual Sheet Forming

Hamid Mozafari[1*], Mojtaba Moravej[1], Mahdi Karami Khorramabadi[2]
[1]Department of Mechanical Engineering, Faculty of Engineering, Payame Noor University, Tehran, Iran
[2]Department of Mechanical Engineering, Khorramabad Branch, Islamic Azad University, Khorramabad, Iran
[*]Email of the Corresponding Author: Mozafari.h@pnu.ac.ir
Received: October, 31, 2024; Accepted: December 26, 2024

Abstract
This article presents a finite element and experimental analysis of the parameters influencing the shaping of the human face profile during the single-point gradual forming process of sheet metal. Single-point progressive forming is a non-traditional technique widely utilized in rapid prototyping and custom part manufacturing. A significant challenge in this process is achieving a greater wall angle while maintaining sufficient depth. The primary objective of this research is to identify and investigate the key factors affecting the accuracy and quality of the forming process. Using finite element analysis software, we conducted detailed modeling of the human face profile and examined the effects of parameters such as sheet material, thickness, and applied forces on the shaping process. The experimental results from practical tests were compared and analyzed against the numerical simulations. This study demonstrates that optimizing these parameters can significantly enhance the quality and accuracy of human face profile shaping, thereby providing a foundation for advancements in the design and production of complex parts. The findings serve as a valuable reference for engineers and designers in industries related to sheet forming and human profile design.

Keywords
Finite Element Method (FEM), Gradual Forming Process, Experimental Testing, Tool Path Optimization

1. Introduction

Sheet forming is a critical process in materials engineering and production that has garnered significant attention since the early 20th century. As technology advances and the demand for complex, lightweight components increases, various sheet-forming methods have emerged, including pressure forming, hot and cold mold forming, and gradual forming [1-2]. The gradual forming process, in particular, has gained prominence due to its capability to produce parts with intricate shapes and high precision. Over the past decade, this method has seen substantial advancements in producing automotive components such as body parts, hoods, and fenders, establishing sheet forming as a key technique across various industries, including automotive, aerospace, and medical product design [3-4].

Recently, shaping human profiles, especially facial profiles, has emerged as a new challenge in this domain. This area has gained particular importance due to its diverse applications in medical device design, robotics, and entertainment. For instance, in developing prosthetics and medical equipment, facial profile shaping accuracy can significantly influence patients' effectiveness and comfort [5].

Finite Element Analysis (FEM) is a powerful tool widely recognized for simulating and investigating the behavior of materials during forming processes. This method enables engineers to predict material behavior under various conditions, significantly reducing the need for costly and time-consuming physical tests. However, achieving accurate results through FEM requires a comprehensive understanding of the parameters that influence the forming process [6].

Historically, early research in this field primarily focused on experimental and observational analyses. With advancements in simulation technologies, FEM has evolved into an essential tool for understanding complex forming behaviors. For instance, a study by Adams et al. [7] in 2015 highlighted the impact of various parameters, such as material type and loading conditions, on the quality of forming intricate profiles. Similarly, Kim et al. [8] 2018 examined the effects of sheet thickness and material type on facial profile shaping, providing significant insights into optimizing shaping conditions.

These studies underscore that variations in process parameters can directly influence the accuracy and final quality of formed profiles. Furthermore, some researchers have explored the effects of environmental conditions, including temperature, on sheet forming. Gupta et al. [9] demonstrated in their 2018 research that temperature fluctuations could significantly alter the mechanical properties of sheet materials, thereby affecting their forming behavior. Additionally, Wang et al. [10] introduced an innovative spiral path strategy by interpolating and translating points generated from Unigraphics software, which further enhances the precision and efficiency of the forming process.

The integration of FEM with experimental data not only facilitates a deeper understanding of the forming process but also paves the way for more efficient design and production methodologies in various applications, including automotive, aerospace, and medical device manufacturing. As the demand for complex geometries and high-performance materials continues to grow, the role of FEM in optimizing forming processes will become increasingly vital.

In this article, we present an experimental and numerical investigation of the parameters influencing the shaping of the human face profile during the single-point gradual forming process of sheet metal. This research identifies and analyzes key factors, including sheet type and thickness, applied forces, and environmental conditions. The findings from this study will contribute to optimizing the shaping process and enhancing the quality of final products while laying the groundwork for future advancements in the design and production of complex components. By integrating experimental results with numerical simulations, this research seeks to deepen our understanding of the forming process and offer optimal solutions for designing and producing human profiles. This approach is a crucial link between theoretical insights and practical applications in materials engineering and complex product design.

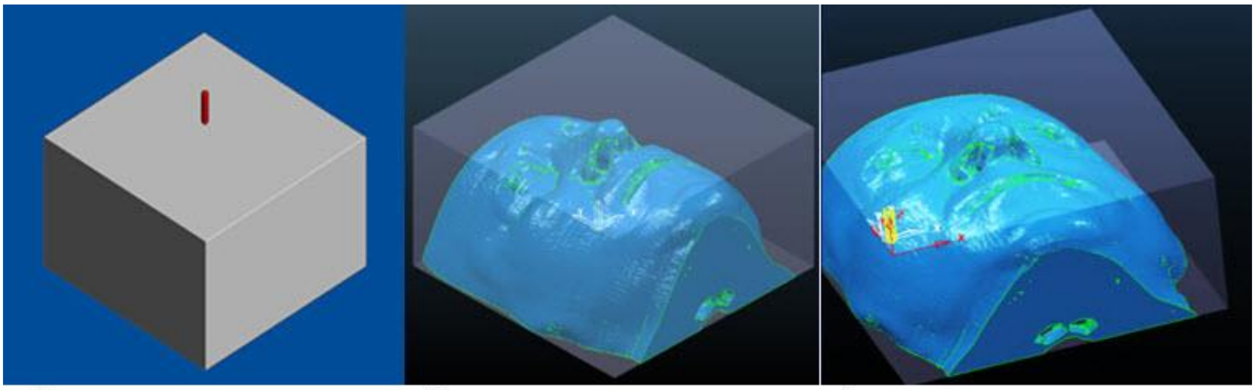
2. Finite Element Analysis

This section discusses finite element analysis (FEM), a crucial tool for simulating and analyzing the human face profile-forming process during the single-point gradual forming of sheet metal. FEM enables accurate modeling of material behavior and the influence of various parameters on the forming process.

The modeling of the gradual forming process was carried out in four distinct stages. First, individual components such as blanks, tools, clamp plates, and back plates were modeled and assembled in 3D using Mimics software. In the second stage, this model was imported into finite element simulation software, where meshing, boundary conditions, material properties, and contact interactions between parts were defined. The third stage involved generating CNC code to create a displacement curve for shaping, followed by the final stage of solution and post-processing.

Initially, the geometry of the human face profile was accurately modeled using Mimics software based on 3D data obtained from medical MRI scans. Precision in geometry modeling is critical, as any errors at this stage can significantly impact the final results. To achieve a more uniform thickness distribution in the sheet, the shaping strategy and programming were explored in Powermill software.

Figure 1a illustrates a block representing the human face piece defined within a bounding box and tools of varying diameters for the shaping process. Figure 1b shows the model derived from the three-dimensional MRI data. In contrast, Figure 1c depicts the tool path's optimization, highlighting the defined block's reference geometry and the working planes within the software environment. Various tool path methods were investigated in the TPO section of Powermill software to enhance tool path optimization.



a) b) c)

Figure 1. a) View of the defined raw block, b) Modeling of the human face based on MRI data, c) Optimization of the tool path on the workpiece

Figure 2a illustrates the top tool's movement path across the workpiece's raw surface. At this stage, the software defines the zero point of the tool, initiating its movement. As the tool traverses the defined raw block, it generates the necessary outputs, specifically the shaping codes required for the process. This precise movement ensures that the shaping aligns with the intended design specifications.

Ultimately, the gradual shaping method applied in the Powermill software allows the final model of the human face profile to achieve the desired shape, as depicted in Figure 2b. This process not only emphasizes the accuracy of tool path programming but also highlights the effectiveness of the gradual shaping technique in producing complex geometries. Integrating these tools and methods ensures that the final part meets the required dimensional and aesthetic standards, paving the way for successful prosthetics and facial reconstruction applications.

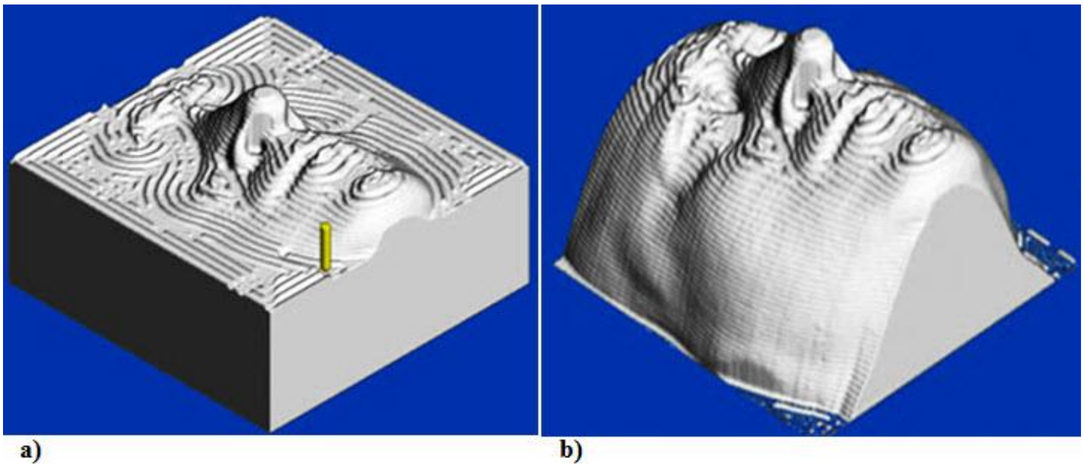


Figure 2. a) Trajectory of the spherical head tool, b) Final part produced representing the human face profile

2.1 Geometrical Modeling of Tools

In this research, the modeling of tools, clamps, and sheets is conducted, assuming all components are completely rigid, as illustrated in Figure 3a. This rigid modeling approach simplifies the analysis and allows for a clearer understanding of the forming process. The tool movement path for shaping is depicted in Figure 3b. As shown in the figure, the tool initiates its movement from point A, advancing along the predetermined path designed for shaping the nose. As it progresses, the tool penetrates deeply into the copper sheet until it reaches point B. The tool then transitions to a transverse movement, advancing to point C. After reaching point C, the second shaping step commences, with the tool moving toward point D. This sequence of movements is repeated in each iteration until the nose piece is fully formed into its final shape.

The entire forming process is conducted at room temperature to maintain material properties. The contact area is lubricated with standard oil to mitigate friction and wear between the tool and the copper sheet. The absence of lubrication can lead to increased wear and chipping of the sheet surface, ultimately compromising the quality of the final product. Additionally, inadequate lubrication results in a higher forming force, which can negatively impact the efficiency and precision of the shaping process. Proper lubrication is, therefore, essential for achieving optimal results in forming complex geometries.

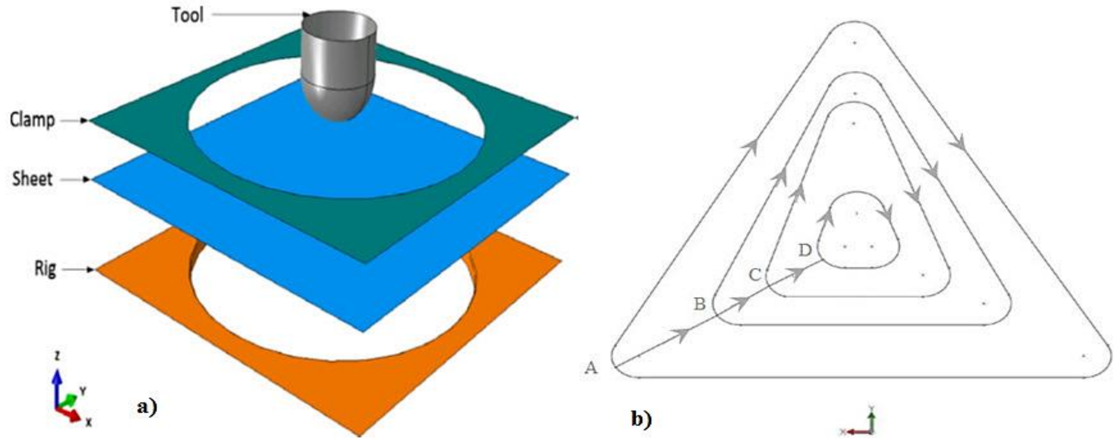


Figure 3. a) 3D modeling of the tool, fixture, and sheet for the nose piece, b) Movement path of the tool for shaping

An explicit dynamic solution code has been employed since the process is conducted at room temperature and involves significant material deformation. This approach allows for the accurate simulation of each phase of the tool's advancement, including its penetration into the sheet and subsequent transverse movements. Specifically, one step has been defined for each stage of the tool's operation to ensure precise tracking of the dynamic changes occurring during the forming process.
In the numerical analysis conducted in this research, several critical variables were examined, including displacement, element rotation, and variations in the thickness of the forming sheet. By analyzing these factors, we can gain insights into the material behavior under the specific conditions of cold forming. Understanding how displacement and rotation affect the overall shaping process is essential for optimizing the tool path and enhancing the quality of the final product. Furthermore, monitoring changes in sheet thickness is crucial for ensuring uniformity and integrity in the formed components, ultimately leading to improved performance in practical applications.

2.2 Material properties

Defining the properties of the material determines its behavior. Determining the mechanical behavior is also one of the most fundamental parts of the mechanical analysis of finite elements, based on which the condition of the material in terms of stress, strain, and other properties is considered. This section determines critical mechanical properties such as elastic behavior, plastic behavior and other mechanical properties of the material. Equation (1) has been used to express the characteristics of a material that undergoes large dynamic deformations.

$$\sigma = [A + B\varepsilon^n] \tag{1}$$

Where σ equals the silane stress, A is the initial yield strength at room temperature, and B and n are the coefficients obtained from the work-hardness values. Table 1 shows the values entered in the equation in the Abaqus software's definition of material properties section. Also, the mechanical characteristics of copper sheets are shown in Table 2.

Table 1. Copper sheet modeling parameters

n	B (Mpa)	A (Mpa)
0.54	426	275

Table 2. Mechanical properties of copper sheet

Properties	Values
Young's modulus	128 (Gpa)
Poisson's ratio	0.33
Yield stress	140 (Mpa)
Ultimate strength	515 (Mpa)
density	8800 (kg/m3)

2.3 Boundary Conditions

After applying the loads to the analysis setup, defining the boundary conditions to restrict the degrees of freedom within the model is essential. In this section, the imposed restrictions limit all degrees of freedom around the nose plate and the back surface, which is crucial for enhancing the formability of the sheet. Additionally, the behavior of the clamp, the sheet, and the overall boundary conditions of the model are clearly outlined. Considering boundary conditions is vital for successfully implementing this forming process. Specifically, the design minimizes any potential movement of the sheet on the clamp, thereby preventing issues such as slipping or unintended deformation. Ensuring appropriate yielding conditions significantly reduces the risk of sheet tearing.

The movement of the sheet is constrained to the Z direction, allowing vertical displacement while restricting lateral movement. This configuration ensures that the sheet remains securely positioned during the forming process. Conversely, the shaping tool is fixed in all degrees of freedom except for its movement in the Z direction, allowing it to advance and penetrate the sheet as needed. Additionally, the tool can move along the X-axis during its transverse advancement, facilitating effective shaping while maintaining stability. These carefully defined boundary conditions play a crucial role in achieving accurate simulation results and ensuring the integrity of the forming process.

3. Experimental Work

The initial step in producing the profile piece of the human face involves preparing a copper sheet with a thickness of 1 mm. This sheet is then cut into a rectangular shape using a guillotine machine, with dimensions measuring 170 x 235 mm. Given the intricate geometry of the human face, which features various angles and a complex structure, a range of shaping tools with diameters varying from 1 to 15 mm is employed. These tools, crafted from ordinary steel, are specifically chosen for their durability and effectiveness in gradually shaping the copper sheet.

The tools are securely fixed by tool holders on the milling machine, ensuring they remain perpendicular to the sheet throughout the shaping process. This alignment is critical for achieving precise contours and details in the final profile. Using different tool diameters allows for a stepwise approach to shaping, enabling gradual refinement of the features of the human face profile.

Figure 4 illustrates the forming tools utilized in this experimental setup, showcasing the variety of tools employed to achieve the desired complexity and accuracy in the final product. This methodical approach enhances the quality of the profile piece and demonstrates the effectiveness of using multiple tools to adapt to the varying geometries encountered in facial modeling.

Figure 4. Forming tools of the tested sample

For this reason, the selected sheet is made of copper, which offers superior strength and significant resistance to corrosion and deformation compared to aluminum during the testing process. The inherent properties of copper make it an ideal choice for applications requiring durability and reliability under mechanical stress.

The copper sheet is securely attached to the milling machine's table using a specialized fixture designed to hold the sheet in place. This fixture ensures that the sheet remains stable throughout the shaping process. The milling machine features a tool holder plate capable of accommodating 24 tools within its compartment, allowing for versatile and efficient operation. As the tool holder plate rotates, the corresponding machine tool engages with the copper sheet, facilitating the gradual shaping required for the profile piece.

To further enhance the stability of the sheet during the progressive forming process, a fixture with six-finger covers is employed, as depicted in Figure 5. This design provides additional support and minimizes any potential movement of the sheet, ensuring precision in the shaping operations. The combination of the robust copper material and the effective clamping mechanism contributes to the overall success of the experimental work.

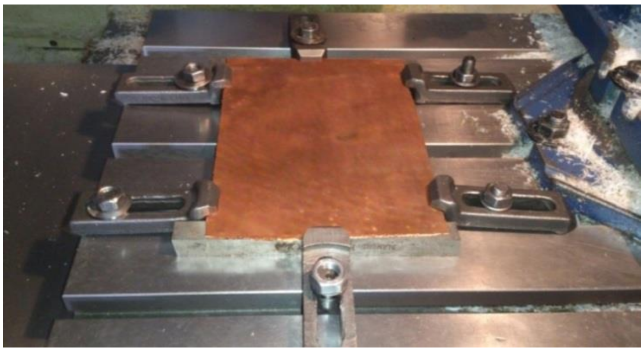


Figure 5. Fixture holding the copper sheet of the test sample

The milling machine utilized in this process is the VMC 850, manufactured by Tabriz Machining Company. Once the copper sheet is placed on the finger covers of the fixture, the fixture itself is secured to the milling machine's table. The shaping tool is then attached to the tool holder, as

illustrated in Figure 6a. During the shaping process, the tool is positioned perpendicular to the sheet, ensuring optimal engagement with the material. After the tool is correctly aligned, it descends to meet the reference table of the milling machine.
The tool begins to rotate at a predetermined speed, altering the shape of the copper sheet through a combination of circular and linear movements. The movement path is meticulously programmed using Powermill software, which defines the zero point of the workpiece. The process initiates from the right corner of the modeled forehead, where the tool penetrates the sheet with a vertical step size. Following this initial penetration, the tool advances transversely across the sheet, gradually shaping the copper into the profile of the human face.
To create the desired facial profile, the first forming marks appear on the copper sheet as the tool applies the vertical load to its surface. Subsequently, through a series of circular and linear motions, the tool meticulously carves out the intricate details of the human face, progressively refining the profile with each pass. Figure 6b illustrates the tool's movement over the sheet during this shaping process, highlighting the specific complexities of producing this detailed component.

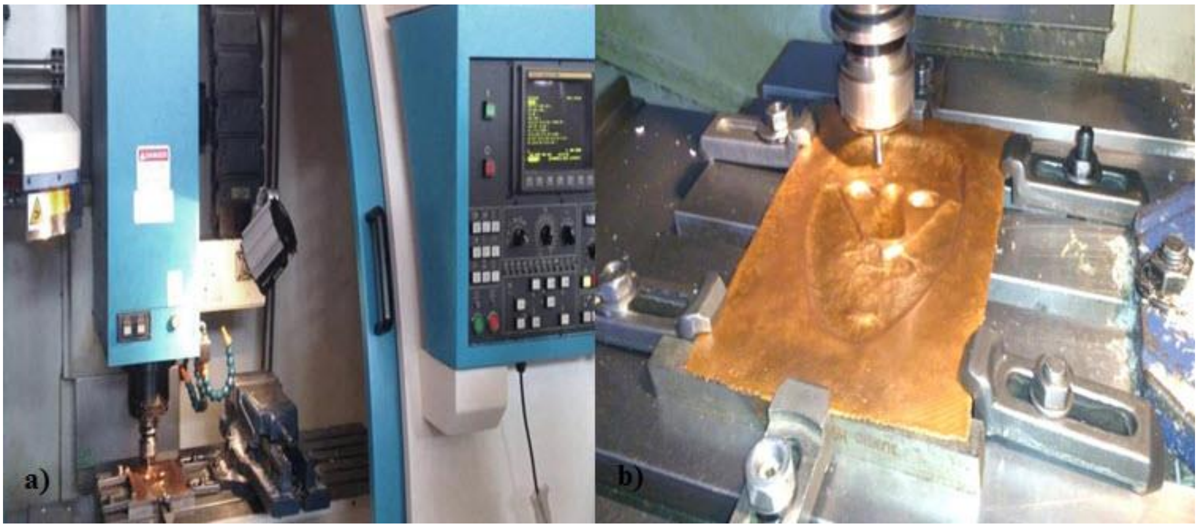


Figure 6. a) Fixture and tool mounted on the milling machine; b) Movement of the tool over the shape on the sheet for the process of shaping the human face

Finally, the component created using the gradual forming method exhibits a maximum depth of deformation of 0.7 mm, specifically at the most contoured area, the tip of the nose. This depth was achieved following the simulation conditions and settings applied within the Powermill software. The meticulous programming and simulation allowed for precise control over the shaping process, ensuring that the final product accurately reflects the intended design.
Figure 7 presents the completed profile of a human face, showcasing the intricate details achieved through this process. The image illustrates the successful outcome following the coding phase, where the shaping instructions were generated using Powermill software and subsequently transferred to the CNC milling machine. This seamless transition from digital design to physical production underscores the effectiveness of modern machining techniques in creating complex geometries.

Figure 7. Generated profile fragment of the human face

4. Results and Discussion

Following the completion of the initial simulation stages and problem-solving processes, the results of the gradual forming of the sheet were analyzed using Abaqus software. To ensure the accuracy of the simulation, these results were compared with experimental tests. The numerical analysis focused on the shaping process of the nose region of the human face, which presents greater complexity and more considerable relative deformation compared to other facial areas.

In this numerical solution, a homogeneous and isotropic sheet was considered. The simulation results were visualized within the software, allowing for a comprehensive examination of various parameters, including stress distribution, plastic strain, equivalent plastic strain, and linear and rotational displacements. This multifaceted analysis provided insights into the mechanical behavior of the material under the specified conditions.

It is important to note that the model developed in this research did not incorporate an analysis of heat distribution within the workpiece. This omission was based on two key observations: firstly, the temperature between the forming tool and the sheet remained relatively stable at room temperature throughout the shaping process; secondly, a sufficient amount of lubricant was present between the tool and the sheet during the experimental work, effectively minimizing wear and thermal effects during the operation.

A tensile test was conducted on the sample to understand the mechanical properties of the sheet further. Figure 8 illustrates the stress-strain curve obtained from the tensile test of the copper sample sheet, providing critical insights into the material's mechanical characteristics. This data validates the simulation results and enhances the understanding of the material's behavior under the specific conditions of the shaping process. Overall, the combination of numerical simulations and experimental validation underscores the effectiveness of using Abaqus for analyzing complex forming processes, particularly in regions of intricate geometry such as the human face. Future work could explore the inclusion of thermal effects and the impact of varying lubrication conditions to refine the simulation outcomes' accuracy further.

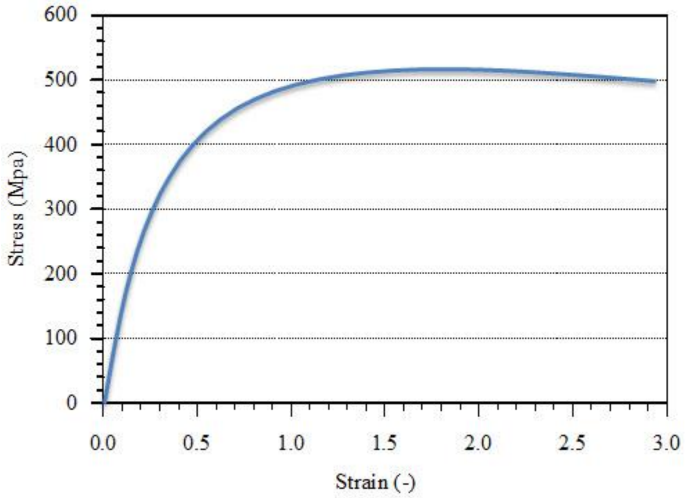


Figure 8. Stress-strain curve from the tensile test of the copper sample sheet

The changes in the thickness of the nose piece, which had an initial thickness of 1 mm, are illustrated in Figure 9a. The simulation results indicate that as the tool moves downward and the forming depth increases, the thickness of the sheet progressively decreases. Notably, the minimum recorded thickness is 0.23 mm, which occurs at the tip of the nose piece. This significant reduction in thickness highlights the material's response to the forming process and the effectiveness of the simulation in capturing these changes. Figure 9b presents the thickness measurement path utilized in the simulation, visually representing how thickness was assessed throughout the forming process. This path is critical for understanding the thickness distribution across the nose piece, as it identifies areas experiencing the most substantial deformation.

The observed thickness reduction can be attributed to several factors, including the material properties, the geometry of the forming tool, and the applied forces during the shaping process. The simulation effectively captures these dynamics, offering valuable insights into the material behavior under the specific conditions of the nose piece forming. Furthermore, the results emphasize the importance of monitoring thickness variations in applications requiring precise geometrical features, such as facial prosthetics or cosmetic implants. The data obtained from this simulation can guide further refinements in the forming process, ensuring optimal material utilization and structural integrity in the final product. Overall, the findings underscore the capability of the simulation to accurately predict thickness changes, which is crucial for both theoretical understanding and practical applications in the field of material forming.

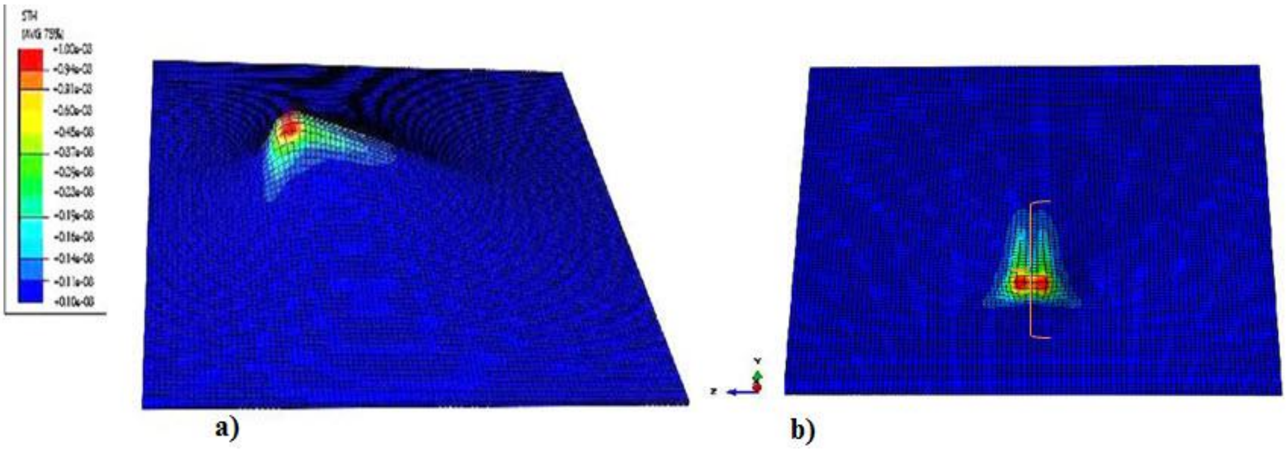


Figure 9. a) Changes in thickness of the nose piece; b) Thickness measurement path in the simulation

A section of the nose was cut, as depicted in Figure 10a, to verify the thickness distribution of the nose piece experimentally. The measurement process commenced from the initial thickness of the sheet and continued along the contour of the nose until reaching the starting point on the opposite side of the sheet.

The results of the thickness measurements for the cut section are presented in **Figure 10b**. The data reveals a clear trend in thickness reduction: starting from the initial thickness of 1 mm, the thickness decreases to 0.77 mm at a distance of 1 mm from the starting point. Beyond this point, the thickness reduction continues at a more gradual rate, resulting in a constant slope along the measured surface until the end of the diagram.

The minimum thickness recorded during the experimental tests was 0.19 mm, indicating a significant reduction in material thickness due to the forming process. This experimental validation aligns well with the simulation results, confirming the accuracy of the numerical model in predicting thickness changes during the shaping of the nose piece. Overall, these findings highlight the effectiveness of both the experimental and simulation approaches in understanding the thickness distribution of complex geometries. The data obtained can be instrumental in optimizing future designs and processes, ensuring that the desired mechanical properties and structural integrity are maintained in the final product.

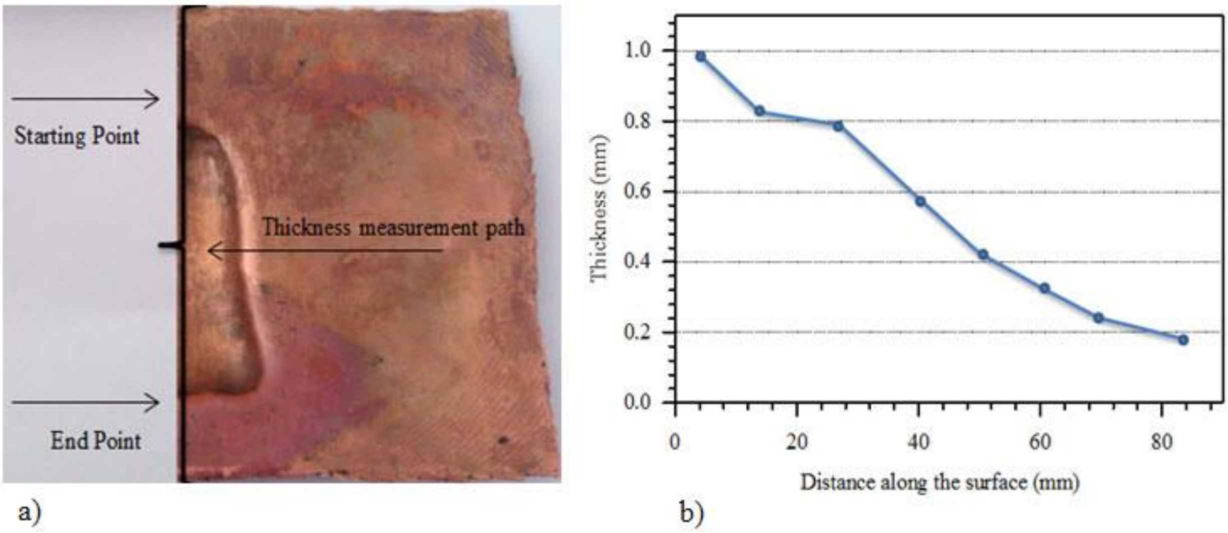


Figure 10. a) Section of the nose piece of the test sample, b) Nose thickness distribution in the IF process (t_0=1mm)

The comparison of thickness distribution among the measured sections, simulation results, and experimental data is presented in Table 3 and illustrated in Figure 11. It is important to note that the distance between the measured points along the cross-section of the nose is 10 mm. The results indicate that the error between the finite element simulation and the experimental measurements is approximately 10%. This discrepancy is considered acceptable and underscores the reliability of the simulation process. The close alignment of the results demonstrates the effectiveness of the criteria employed to assess the mechanical behavior of the sheet during the forming process.

Table 3 provides a detailed comparison of the thickness distribution obtained through both the finite element method and the experimental approach. This side-by-side analysis validates the simulation results and highlights the nuances of material behavior under different conditions. The findings reinforce the significance of using numerical simulations in conjunction with experimental methods to achieve a comprehensive understanding of material deformation. The 10% error margin suggests that the finite element analysis can be a valuable tool for predicting thickness distribution in similar forming processes, paving the way for improved design and optimization in future applications.

Table 3. Comparison of sheet thickness distribution by finite element method and experimental method

Experimental method (mm)	Finite element method (mm)
1	**1**
0.83	**0.94**
0.79	**0.81**
0.58	**0.60**
0.42	**0.45**
0.33	**0.37**
0.24	**0.29**
0.19	**0.23**

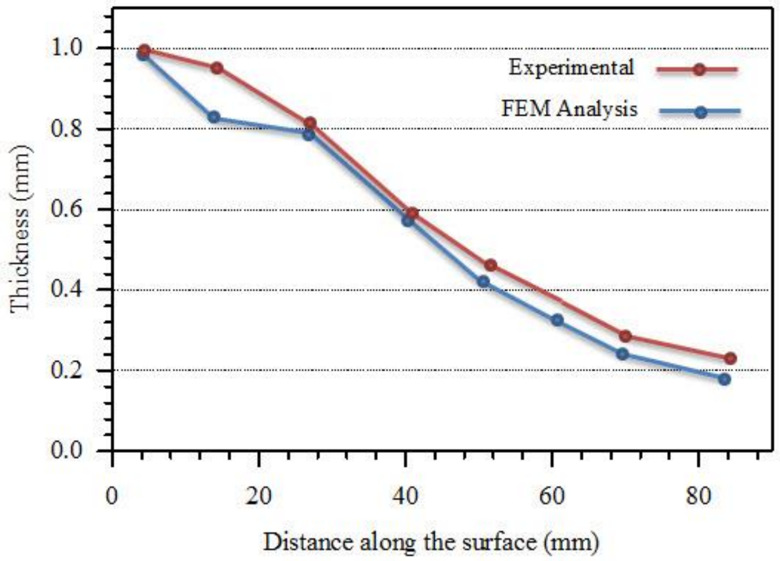


Figure 11. Thickness distribution obtained from experimental tests and simulations of the test piece

As illustrated in Figure 12a, the practical results obtained from the meshing of the nose piece reveal that, with an increase in forming depth and subsequent stretching of the sheet, the displacement of the elements increases significantly. This phenomenon indicates that the elements are moving apart from each other due to the applied forces during the forming process. Notably, the maximum linear displacement occurs at the tip of the nose piece, reaching a value of 1.285 mm. This substantial displacement underscores the extent of deformation experienced by the material in this region.

Figure 12b presents the rotational displacement of the elements in the nose piece. Similar to the linear displacement, the maximum rotational movement is also observed at the tip of the nose piece, with a value of 1.135 degrees. This rotational displacement is crucial for understanding how the geometry of the nose piece changes during the forming process. It can affect the overall fit and function of the final product.

The data from both linear and rotational displacements provide valuable insights into the mechanical behavior of the nose piece under the specified forming conditions. These findings confirm the simulation's effectiveness in capturing the complex deformations and highlight the critical areas where the material undergoes the most significant changes. Overall, the results emphasize the importance of monitoring linear and rotational displacements in designing and optimizing forming processes, particularly for intricate geometries such as facial components. This comprehensive understanding can guide future developments in material forming techniques, ensuring better performance and accuracy in the final applications.

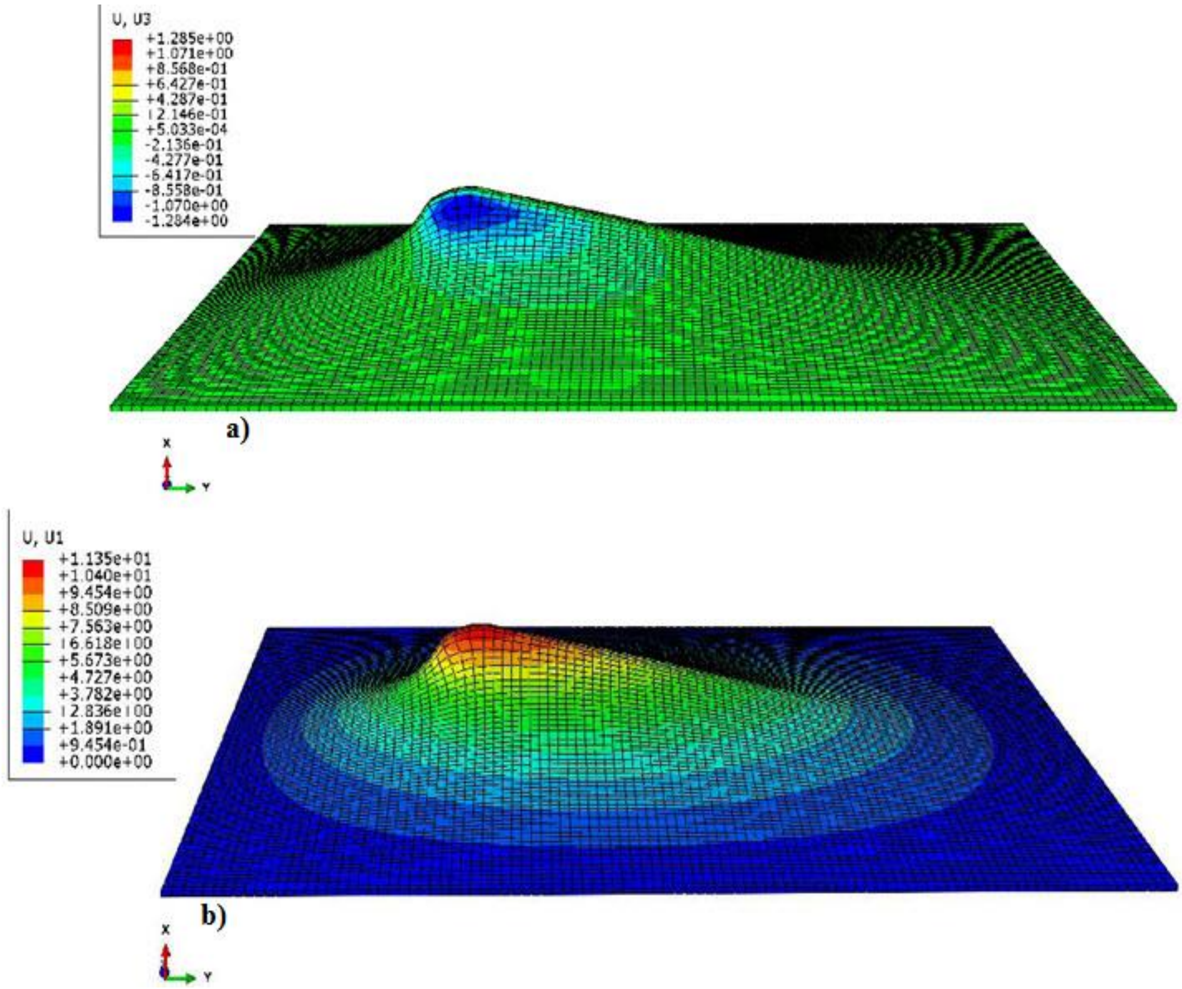


Figure 12. a) Spatial displacement of the nose piece; b) Rotational displacement of the nose piece of the test sample

The copper sheet was meticulously meshed before the forming process to evaluate the accuracy of the simulation results against the experimental tests regarding the displacement of elements and linear strain. Following the forming operation, the linear strains were calculated based on the displacement data obtained from both the simulation and experimental tests. These calculated values were then systematically compared and presented in Figure 13. The comparison illustrated in Figure 13 highlights the correlation between the simulation results and the experimental measurements for the nose piece. By analyzing the linear strains, we can observe how closely the simulation predicts the actual behavior of the material during the forming process. The alignment of these results validates the finite element model used in the simulations, indicating that it accurately captures the key mechanical responses of the copper sheet under the specified conditions.

This comparative analysis not only reinforces the reliability of the simulation approach but also provides insights into any discrepancies that may exist, which can be critical for refining the modeling techniques in future studies. Understanding these differences is essential for optimizing the forming process and ensuring the final product meets the desired specifications. Overall, the findings from this analysis contribute significantly to the body of knowledge regarding the behavior of materials during forming processes and underscore the importance of integrating both simulation and experimental methodologies for comprehensive assessments.

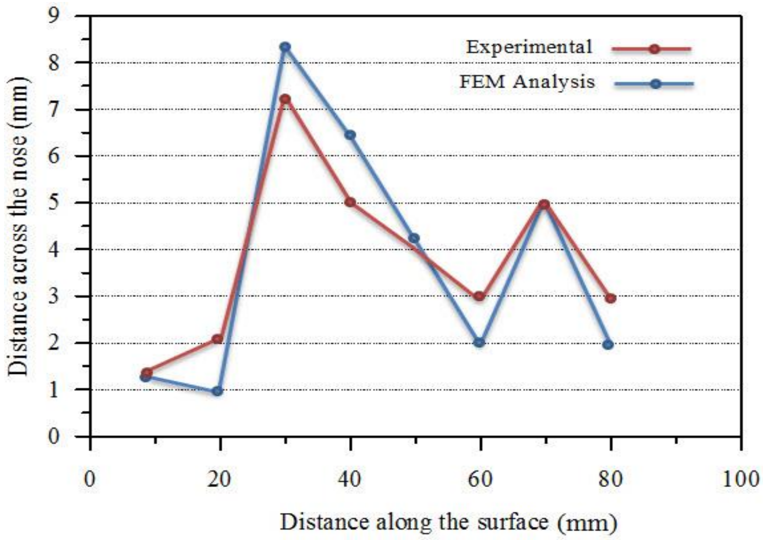


Figure 13. Comparison of displacement results obtained from simulation and experimental tests for the nose piece

The numerical and experimental results for the limit forming angles using different tool diameters are presented in Figures 14a and 14b. The average difference between the experimental and numerical methods is approximately 2.4% for the limit forming angle, which is considered acceptable and indicates a strong correlation between the two approaches. As illustrated in Figures 14a and 14b, a clear trend emerges: smaller tool diameters correspond to higher formability of the sheet. This observation can be attributed to the deformation conditions experienced by the sheet in the contact area with the tool. When larger tool diameters are used, the deformation conditions closely resemble those of an expansion process. This similarity can increase tensile strain in the contact area, raising the risk of material failure. Conversely, reducing the tool diameter concentrates the deformation forces over a smaller area, enhancing the sheet's formability. This localized application of forces allows for more effective control over the material's deformation behavior, resulting in improved performance during the forming process. Overall, these findings underscore the importance of tool geometry in determining the formability of materials. The results validate the numerical simulations and provide practical insights for optimizing tool design in forming operations. Understanding the relationship between tool diameter and formability can improve process control and product quality in various manufacturing applications.

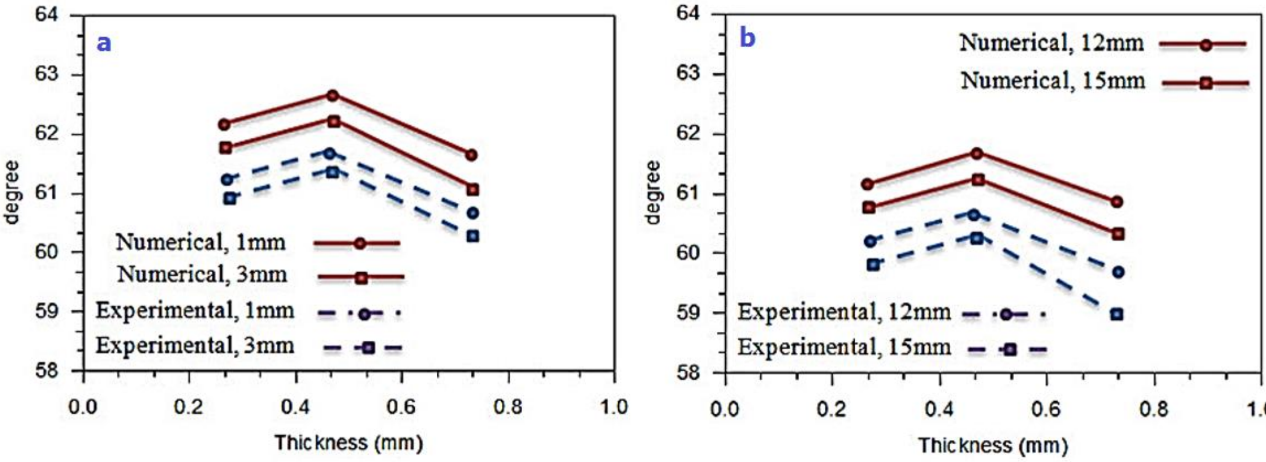


Figure 14. Comparison of the numerical and experimental methods for the limit angle of plasticity: a) Tools with diameters of 1 mm and 3 mm; b) Tools with diameters of 12 mm and 15 mm

5. Conclusion

The advancement of unconventional technologies demands a relentless pursuit of optimal process efficiency. Achieving this goal requires adopting innovative analytical methods to understand better the phenomena involved. This study highlights the integration of the finite element method (FEM) with experimental testing as a powerful approach for analyzing the gradual forming process. Utilizing Abaqus software, we successfully simulated this process to gain deeper insights. Our research included a comprehensive analysis of MRA data, detailing the modeling of an MRA file within the SolidWorks environment and examining human facial images to inform our modeling strategies. Experimental tests were conducted on gradually shaping a copper piece to replicate a human facial profile. This investigation covered the entire production process, including tool selection, the milling machine employed, and the optimization of tool movement paths using Powermill software.

We investigated various parameters to assess the simulation results, including stress types, distribution, plastic strain, and the material's displacement and rotation. Our findings revealed that the maximum stress and strain occur at the contact point between the spherical head tool and the sheet, with values increasing as the forming depth increases. Notably, a forming depth of 0.7 mm yielded the highest stress and strain at the tip of the nose piece. We compared the results with those from experimental work to validate and refine our finite element simulations. The maximum discrepancy between the two was found to be 10%, underscoring the accuracy of our simulations and the robustness of the criteria used to characterize the mechanical behavior of the sheet.

In conclusion, this research demonstrates the efficacy of combining finite element simulations with experimental methodologies to enhance our understanding of material behavior during forming. The results validate the simulation approach and provide critical insights for optimizing manufacturing processes involving complex geometries. Future investigations can build on these findings to further refine techniques and improve the efficiency and quality of unconventional manufacturing technologies, paving the way for advancements in the field.

6. References

[1] Ndip-Agbor, E., Cheng, P., Moser, N., Ehmann, K. and Cao, J. 2019. Prediction of Rigid Body Motion in Multi-pass Single Point Incremental Forming. Journal of Materials Processing Technology. 269: 117-127. doi:10.1016/j.jmatprotec.2019.02.007.

[2] Esmailian, M. and Khalili, K. 2020. Prediction of Tool Force in Two Point Incremental Forming by Slab Analysis. International Journal of Engineering, Transactions B: Applications. 33(11): 2399-2407. doi:10.5829/ije.2020.33.11b.30.

[3] Shakir, G., Hengan, O. and Graham, M. C. 2016. Review on the Influence of Process Parameters in Incremental Sheet Forming. The International Journal of Advanced Manufacturing Technology. 87: 479–499. doi:10.1007/s00170-016-8426-6.

[4] Hou, Y., Myung, D., Park, J. K., Min, J., Lee, H., El-Aty, A. and Lee, M. 2023. A Review of Characterization and Modelling Approaches for Sheet Metal Forming of Lightweight Metallic Materials. Materials. 16: 836-842. doi:10.3390/ma16020836.

[5] Julie, D., White, I., Sahin, N., Ryan, J., Hanne, H. and Jasmien, R. 2021. Insights into the Genetic Architecture of the Human Face. Nature Genetics. 53(1): 45–53. doi:10.1038/s41588-020-00741-7.

[6] Memicoglu, P., Music, O. and Karadogan, C. 2017. Simulation of Incremental Sheet Forming Using Partial Sheet Models. Procedia Engineering, 207: 831-835. doi:10.1016/j.proeng.2017.10.837.

[7] Adams, D. and Jeswiet, J. 2015. Design Rules and Applications of Single-point Incremental Forming. Proceedings of the Institution of Mechanical Engineers Part B: Journal of Engineering Manufacture. 229(5): 754-760. doi:10.1177/0954405414531426.

[8] Kim, H., Park, T., Esmaeilpour, R. and Pourboghrat, F. 2018. Numerical Study of Incremental Sheet Forming Processes. IOP Conference Series: Journal of Physics. 1063:12017. doi:10.1088/1742-6596/1063/1/012017.

[9] Gupta, P. and Jeswiet, J. 2018. Effect of Temperatures during Forming in Single Point Incremental Forming. International Journal of Advanced Manufacturing Technology. 95 (9-12): 3693-3706. doi:10.1007/s00170-017-1400-0.

[10] Wang, J., Li, L., Zhou, P., Wang, X. and Sun, S. 2019. Improving Formability of Sheet Metals in Incremental Forming by Equal Diameter Spiral Tool Path. The International Journal of Advanced Manufacturing Technology. 101(1-4): 225-234. doi:10.1007/s00170-018-2911-z.