Journal of Modern Processes in Manufacturing and Production, Volume 9, No. 2, Spring 2020

DOR: 20.1001.1.27170314.2020.9.4.5.3

Research Paper

Numerical Study of Effective Parameters in the Deep Drawing Process of a Cylindrical Cup and Comparison with Experimental Results

Hamidreza Gharehchahi[1], Mohammad Javad Kazemzadeh-Parsi[2*], Ahmad Afsari[2], Mehrdad Mohammadi[3]

[1]Ph.D. Student, Department of Mechanical Engineering, Shiraz Branch, Islamic Azad University, Shiraz, Iran

[2]Associate Professor, Department of Mechanical Engineering, Shiraz Branch, Islamic Azad University, Shiraz, Iran

[3]Assistant Professor, Department of Mechanical Engineering, Shiraz Branch, Islamic Azad University, Shiraz, Iran

*Corresponding Author: kazemzadeh@iaushiraz.ac.ir

Received: Appril 1, 2020; Accepted: May 27, 2020

Abstract

Today, metal forming is considered one of the essential methods of manufacturing and producing parts. Therefore, the more accurate knowledge of it leads the industrialists to produce higher quality parts. Deep drawing is one of the most important methods in metal forming processes used to produce cup-shaped products. In this paper, numerical simulation of the deep drawing process based on the finite element method is performed using Abaqus software for a cylindrical cup. Then, the results obtained from numerical simulation are compared with the experimental results in the sources, and the validation of the simulation is performed. In the deep drawing process, effective parameters such as circumferential strain distribution, thickness strain distribution, and radial force distribution are extracted from numerical simulations and compared with experimental results in the sources. The effect of friction coefficient, blank holder force, and punch radius on the deep drawing process has also been investigated. Because experimental methods based on trial and error are time-consuming and costly to achieve the shape of the primary blank, researchers use numerical methods to simulate and design metal sheet forming processes such as deep drawing. It is necessary to compare the results with experimental works to validate the simulations performed by numerical methods.

Keywords

Metal Forming, Deep Drawing, Finite Element Method, Numerical Simulation

1. Introduction

Deep drawing is a type of metalworking process used to form flat sheets and turn them into cup-shaped products such as bathtubs, sinks, cups, shell enclosures, and car fenders. In this process, they are first made into the blank and then placed on the die. The desired shape is obtained by pressing the blank into the die utilizing a punch. A compressive force applied to the sheet can prevent the sheet from shrinking when it is pulled into the die. In this regard, pieces of equipment such as a die, punch, and blank holder are needed. The final dimensions of the produced part are a function of the original blank.

In this paper, the deep drawing process simulation based on the finite element method is performed using Abaqus software for a cylindrical cup. Then, the results obtained from numerical simulation are compared with the results of Woo. [1], and the validation of the simulation is investigated. A three-dimensional shell element was used for analysis. All die components are simulated except for the blank as a rigid shell or as a simple geometry as a rigid analytical. The blank is simulated as a deformable shell. The behavior of the material is assumed to be elastic-plastic; however, fracture, formation, and expansion of the crack will not be considered. The problem will be analyzed without considering the dynamic effects. Moreover, the temperature will be constant, and the heat generated will be ignored, but the friction between the workpiece, the die, and the punch will be considered.

The behavior of a phenomenon in a system depends on the geometry of the problem, the properties of the material or environment in which the phenomenon occurs, and the initial boundary conditions and loads present in it. In an engineering system, the geometry of the problem can be very complex. Besides, the initial boundary conditions can be very complicated. Therefore, in general, the analytical solution of the differential equations governing such a system can be very difficult or impossible. In practice, these problems are often solved through numerical methods. Among the existing numerical methods, the finite element method is more comprehensive than other numerical methods due to its efficiency and flexibility.

The finite element method in modeling and simulation of advanced engineering systems in various fields such as construction, transportation, telecommunications, and the like has become one of the key and unavoidable tools [2,3,4]. In building such advanced systems, engineers and designers face precise processes, including modeling, simulation, analysis, design, prototyping, testing, and mass production. Note that to ensure proper operation and cost-effectiveness of the final product, the processes outlined above require much time in the pre-mass production stages of a new and advanced product[5,6]. The process of natural products is often repetitive, meaning that some steps are repeated based on the results obtained in other states and the current state of the design to achieve the best performance at the lowest cost. Therefore, rapid and efficient modeling and simulation techniques are increasingly important in the production of such products. Accordingly, the application of the finite element method in such conditions has been widely used[7].

2. Geometry and specifications of the cylindrical cup

The experimental example selected from Woo paper [1] is a cylindrical container. Figure 1 shows the required die geometry, and Figure 2 shows the disassembly view and how to simulate the die components. The blank is made of copper with a thickness of 0.35 inches. The stress-strain relationship used is shown in Figure 3. The initial blank diameter is 4.44 inches, the blank holder force is 0.5 tons, the die housing diameter is 2.213 inches, the die chamber profile radius is 0.5 inches, the hemisphere diameter is 2 inches, the punch stroke is 1.6 inches, and the friction coefficient is 0.04. [2] The quadratic model is simulated to reduce the computational volume.

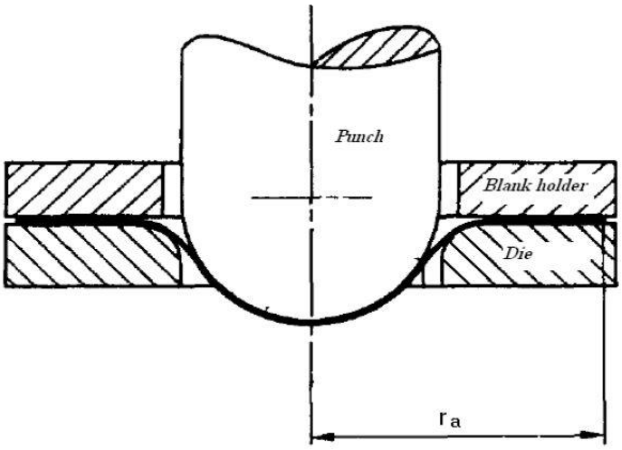

Figure 1. The geometry of forming die for the deep drawing process of the cylindrical cup [1]

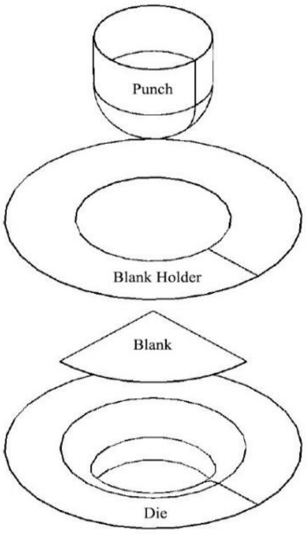

Figure 2. Disassembly view of die components to simulate the deep drawing process of the experimental sample

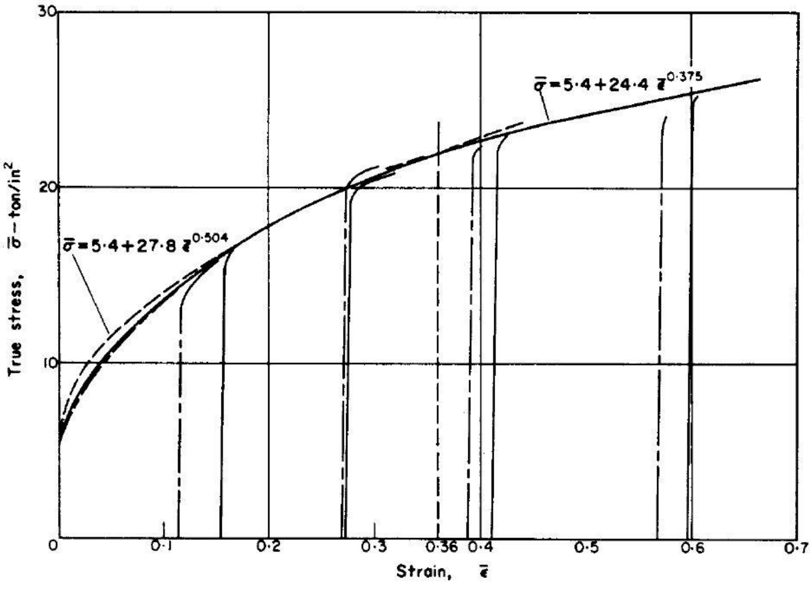

Figure 3. Stress-strain curve of the experimental sample [1]

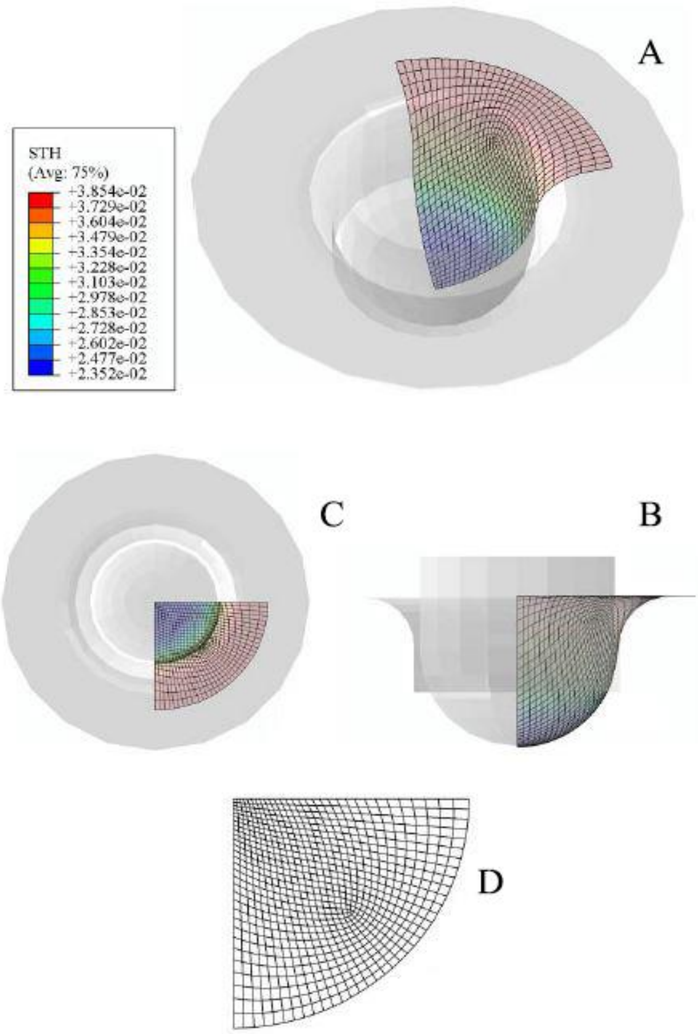

Figure 4. Simulation method, a. Three-dimensional view and thickness distribution, b. Face view, c. Top view, d. Blank element method before deformation

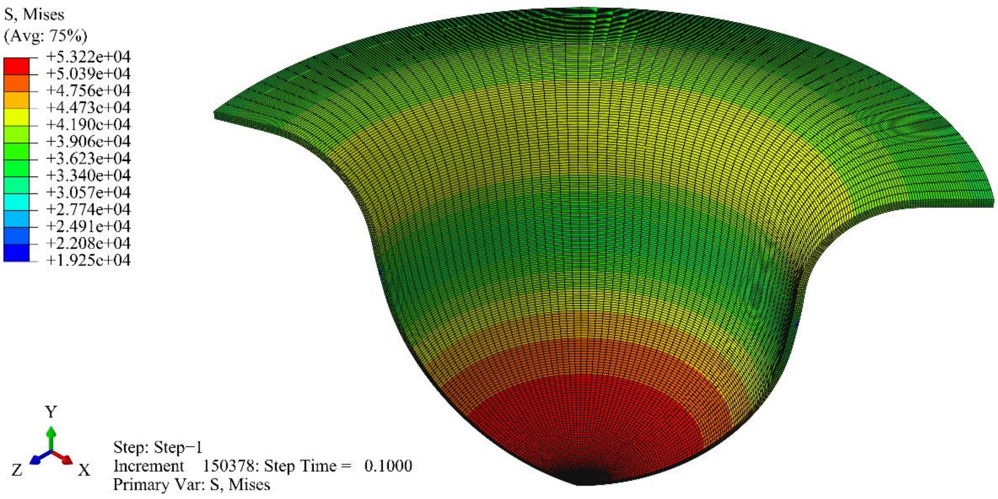

Figure 5. Three-dimensional view of the sheet at the end of the numerical simulation process

Figure 4 shows the simulation method, the three-dimensional view, the distribution of the thickness of the cylindrical cup in the last course of the punch, and how the blank is aligned. A total number of 756 shell elements, including 805 nodes, are used to simulate the process. Figure 5 shows a three-dimensional view of the sheet at the end of the numerical simulation process.

3. Comparison of results

The experimental thickness strain distribution relative to the radius at the moment in Woo paper [1] is obtained as Figure 6. In this figure, Ra is the radius of the initial blank, and ra is the radius of the edge of the blank at the moment, which is also shown in Figure 6. Thickness strain is also defined as Equation 1.

$$Thickness\ strain = \frac{Final\ thickness - Initial\ thickness}{Initial\ thickness} \tag{1}$$

After simulating the process with problem data and extracting the desired results for comparison with different tensile ratios (ra/Ra) are given. Figure 6 shows the experimental thickness strain distribution, and Figure 7 shows the thickness strain distribution obtained from numerical simulations for tensile ratios of 0.82, 0.9, 0.92, 0.95, 0.97.

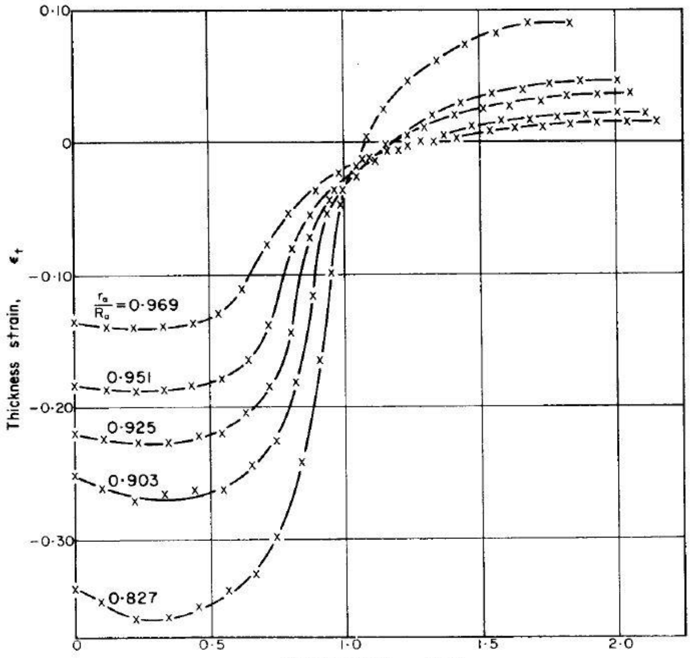

Figure 6. Experimental thickness strain distribution [1]

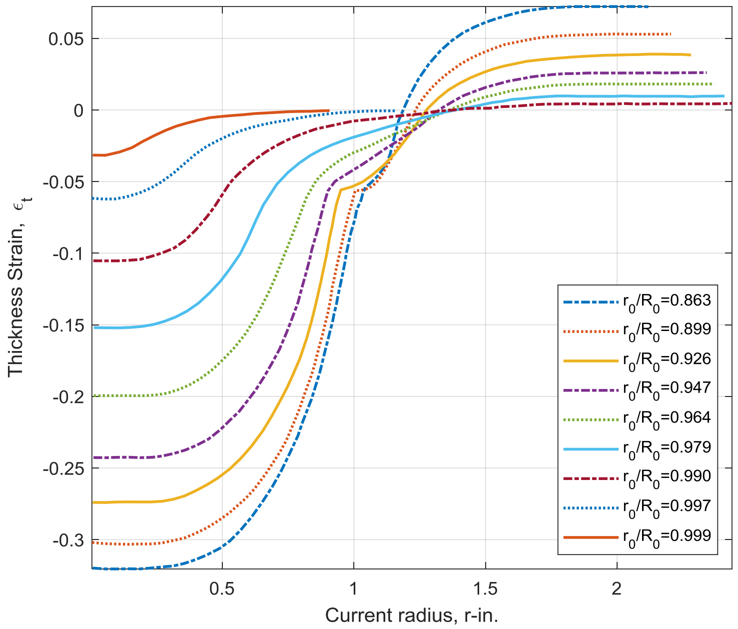

Figure 7. Thickness strain distribution of numerical simulation

Figure 7 shows that the thickness strain is affected by the material in the drawing zone. The strain significantly depends on the friction conditions between the sheet and punch surfaces. For a given deep drawing operation, the lower friction, the greater force transmitted to the drawing region, thus increasing the deformation of the material. However, in a deep-drawing operation, friction in the forming area is desirable to reduce the drawing strength of the material to reduce the load-bearing effect. On the other hand, in a deep-drawing operation that involves only the drawing forming process, the uniform strain of the material is required for deeper presses. In such cases, friction should be minimized by effective lubrication. Figure 8 compares the results of punch load with punch movement in numerical simulation with experimental data.

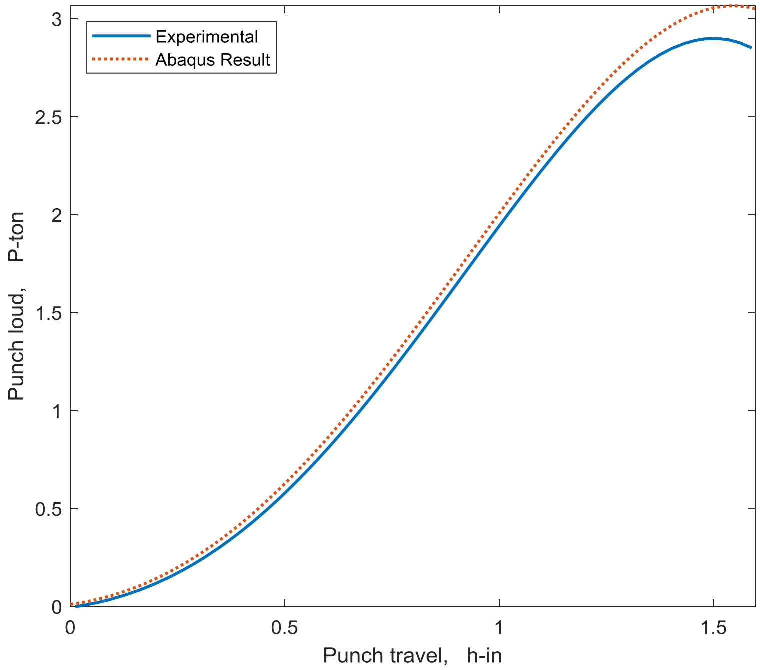

Figure 8. Comparison of punch load results with punch travel in numerical simulation with experimental data

Strains can be calculated at each stage of the deep drawing process, and the analysis of the drawing forming process continues step by step. The results of numerical simulations for the strain history for the boundary elements in terms of the percentage reduction in crude part diameter are shown in Figure 9. They are correct if they match the experimental results shown in Figure 10. A comparison of these two figures shows a good agreement between the simulation results and the experimental results. Figure 11 shows the distribution of circumferential strain in terms of radius from numerical simulation results, which is in good agreement with the experimental results shown in Figure 12.

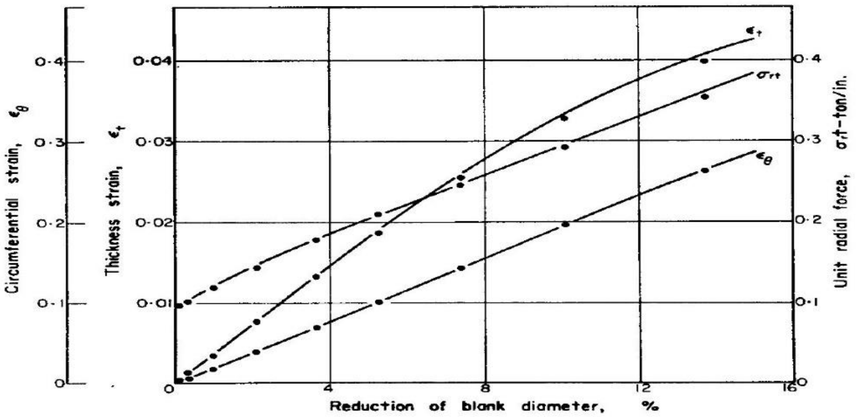

Figure 9. Strain diagram for boundary elements in terms of percentage reduction in raw part diameter obtained from [1]

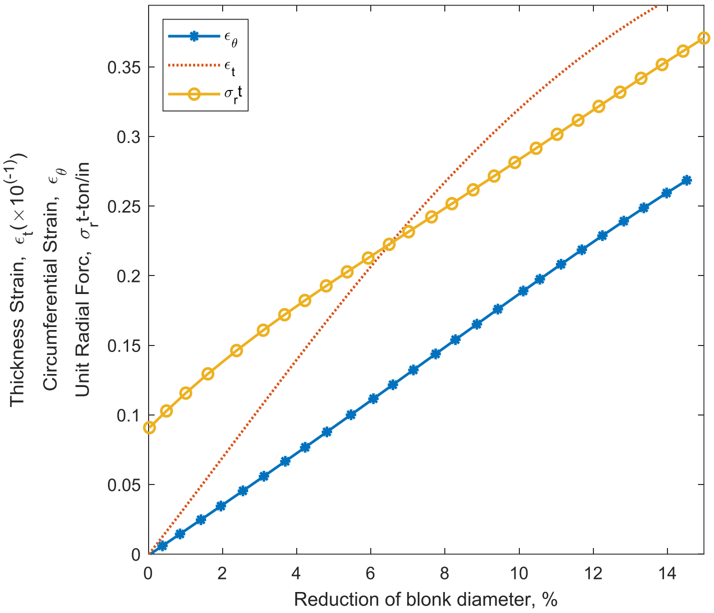

Figure 10. Strain diagram for boundary elements in terms of percentage reduction in raw part diameter in numerical simulation

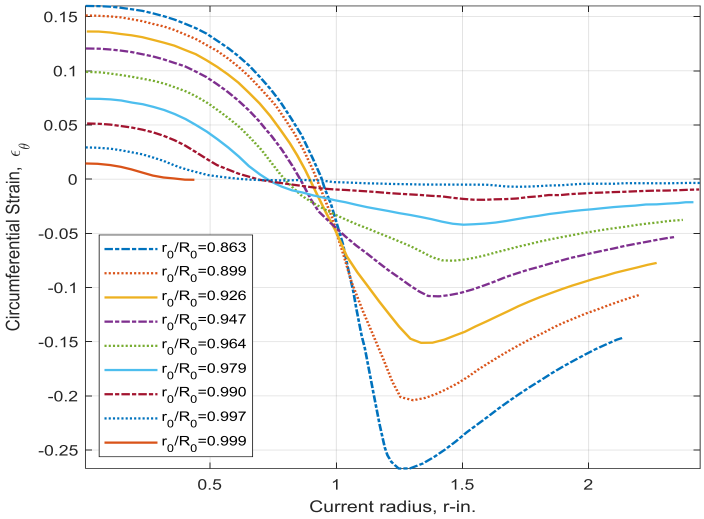

Figure 11. Circumferential strain distribution diagram in terms of radius, obtained from numerical simulation results

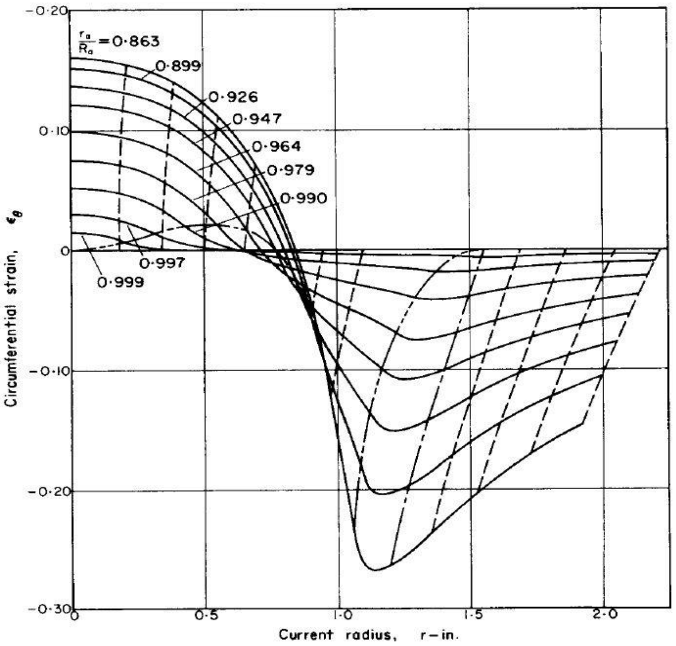

Figure 12. Circumferential strain distribution diagram in terms of radius from experimental results [1]

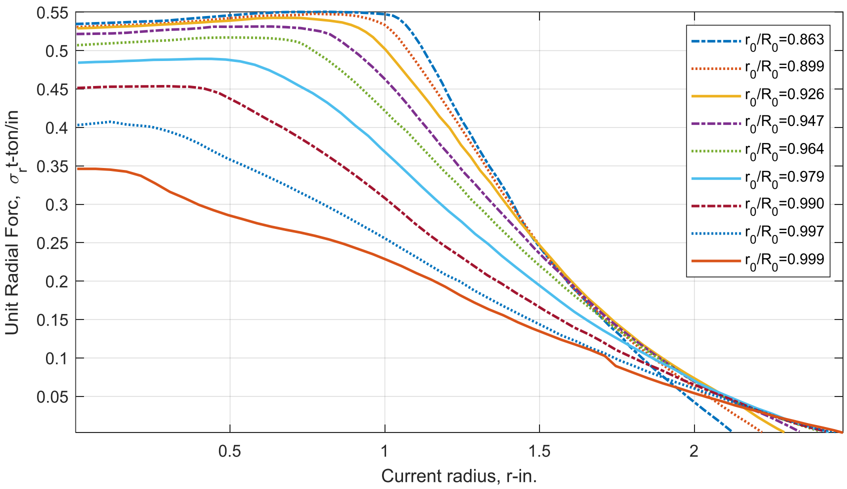

Figure 13. Unit radial force distribution diagram in terms of radius from numerical simulation results

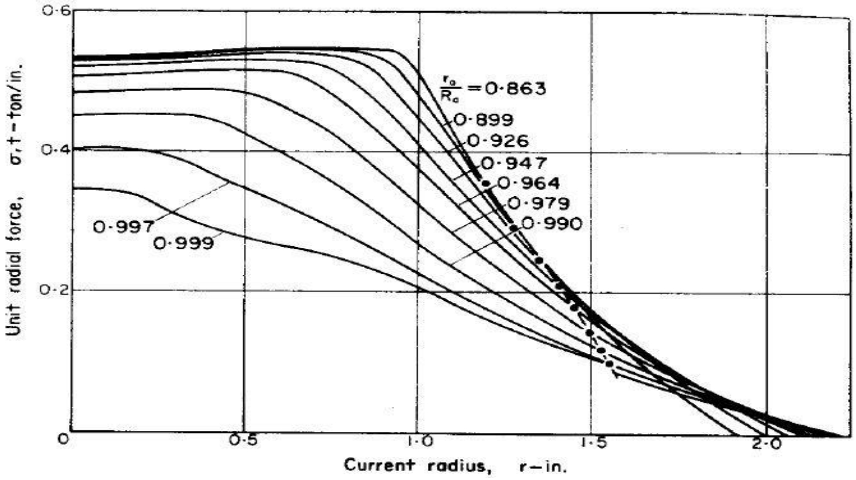

Figure 14. Unit radial force distribution diagram in terms of radius from experimental results [1]

One of the most important parameters in the process of deep drawing is the friction between the parts. Figure 15 shows five different states of friction. According to the diagrams, it can be seen that with increasing the friction coefficient, the force required to produce the desired part increases, but this increase is not significant. Among the diagrams, when the friction coefficient exceeds a certain value, here the number 0.045, the amount of force decreases after reaching a final point. This is due to the gap in the sheet and the decrease in stiffness, resulting in a reduction in force as the process progresses.

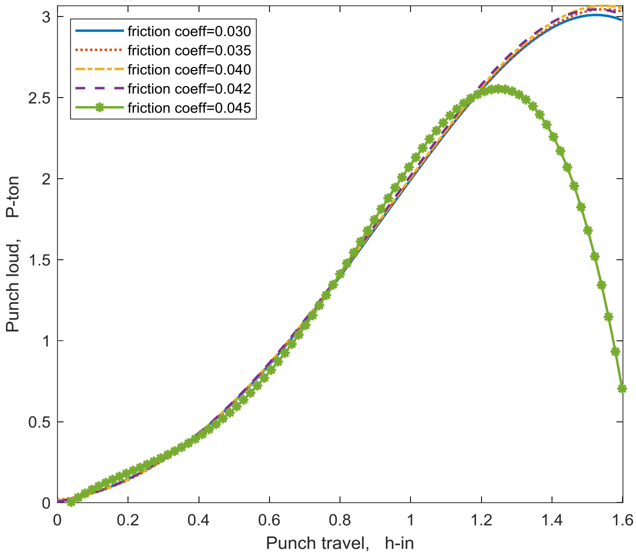

Figure 15. Comparison of simulation results in Abaqus software for variable coefficients of friction

In Figure 16, the simulation results are obtained for changes in the blank holder force. As expected, the force required to move the punch increases with increasing blank holder force. The excessive increase of the blank holder force causes the sheet not to have the necessary freedom to move during the deep drawing process, which by considering the constant displacement for the punch, can cause the sheet to yield to the stress limit and eventually to rupture. This phenomenon has occurred here for the force of 1300 kg, which can be prevented by initial calculations.

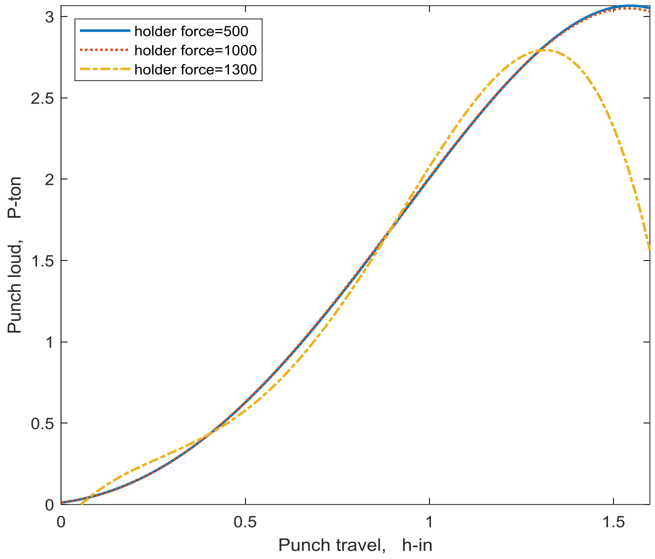

Figure 16. Comparison of simulation results in Abaqus software for different forces for the blank holder

In the next study, by changing the radius of the punch, we tried to determine the effect of this fundamental parameter on the results by considering three different radii for the punch and, consequently, the changes in the matrix, holder, and overall geometry. For the punch in Figure 17, it is shown that according to this diagram, the force decreases with increasing the radius of the punch, and this is due to the constant radius of the sheet because of the smaller the radius of the punch, the smaller the inner radius of the matrix and holder. Considering the external radius of the surface involved between the matrix and the holder is increased, the force required to perform the process is increased.

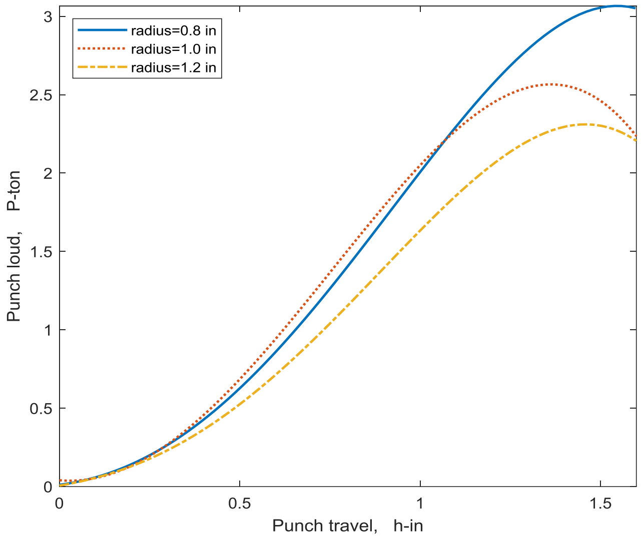

Figure 17. Comparison of simulation results in Abaqus software per punch with different radii

4. Conclusion

In this paper, the deep drawing process of a cylindrical cup is investigated. Modeling of this set was done in Abaqus software, in which the unique axial symmetry feature was used. Axial symmetry can be used when the geometric shape of the problem and the way the whole system is loaded relative to an axis of symmetry can be used in the discussion of the deep drawing of cylindrical parts. After modeling the set was performed entirely in software, the effect of parameters such as circumferential strain distribution, thickness strain distribution, and radial force distribution were investigated and compared with experimental results. In summary, the following can be mentioned as the results of this research.

- Comparing the results of the parameters of circumferential strain distribution, thickness strain distribution, and radial force distribution in numerical simulation with experimental results showed that numerical simulation has good accuracy in deep drawing process analysis.
- The friction coefficient parameter between the parts is very important in the deep drawing process, and if the desired amount of lubrication is not performed, it can cause the sheet to rupture during the process. By increasing the coefficient of friction, the force required to produce the desired part increases.
- If more significant than a certain value, the blank holder force can prevent the part from moving during the process and cause cracks or thinning. Also, if the amount of force is less than the standard value, the sheet may wrinkle at its edge. The force required to move the mandrel increases as the holding force increases.
- The force required to move the punch decreases as the punch radius increases.

5. References

[1] Woo, DM. 1968. On the complete solutions of the deep drawing problem. International Journal of Mechanical Sciences. 10(2):83–94.

[2] Swift, H. W. Plastic bending under tension. Engineering.166:333-359.

[3] Park, S. H., Yoon, J. W., Yang, D. Y. and Kim, Y.H. 1999. Optimum blank design in sheet metal forming by the deformation path iteration method. International Journal of Mechanical Sciences. 41(10):1217-1232.

[4] Son, K. and Shim, H. 2003. Optimal blank shape design using the initial velocity of boundary nodes. Journal of Material Processing Technology. 134(1):92–98.

[5] Hammami, W., Padmanabhan, R., Oliveira, M. C., BelHadjSalah, H., Alves, J. L., Menezeset, L. F. 2009. A deformation based blank design method for formed parts. International Journal of Mechanics and Materials in Design. 5(4):303-314.

[6] Fazli, A. and Arezoo, B. 2012. A comparison of numerical iteration based algorithms in blank optimization. Finite Element in Analysis and Design. 50:207-216.

[7] Itoh, T. 1989. Numerical Techniques for Microwave and Millimeter and Millimeter-Wave Passive Structures. Second Edition, New York: Wiley.