Comparison of Numerical and Experimental Results in Hemispherical Punch Stretching Test

Mevlut Turkoz[a], Osman Yigit[b], Murat Dilmec[c], Huseyin Selcuk Halkaci[c], Behcet Daghan[c]

[a]*Selcuk University, Institute of the Natural and Applied Sciences, Konya, Turkey*
[b]*Karabuk University, Department of Mechanical Engineering, Karabuk, Turkey*
[c]*Selcuk University, Department of Mechanical Engineering, Konya, Turkey*

Abstract. In recent years, considerable effort has been dedicated to the numerical methods capable of modeling sheet metal forming processes. The aim of this effort is to reduce the die try-out period. In this paper, a comparative study was performed between experimentally and numerically obtained strains, sheet thicknesses and part geometries. Different sheet metal samples were clamped between circular die rings and deformed by a hemispherical punch. Before the sheets had been formed, specimens were marked with line patterns grid to measure the deformations after the test. Measuring of the strains and calculation of the thickness was performed with the automated strain analysis system (ASAME). Prediction of the strains was performed with ABAQUS. In addition, geometrical comparison was also performed by measuring dimensions of the specimens with using 3D scanning optical device.

Keywords: Sheet metal forming, Finite Element Method, ABAQUS, ASAME, 3D scanning
PACS: 81

INTRODUCTION

Sheet metal forming simulation is a powerful technique for predicting the formability of parts and has increasingly become an important tool for the process optimization [1,2]. These simulations provide a significant reduction in both cost and time compared with the use of die try-out method that is a very time and cost consuming [2]. However, it is required to experimentally verify simulation results.

During recent years, the computer-aided optical measuring and strain measurement methods are gaining importance more and more. While some optical measuring systems provide dimensions of 3D parts, optical strain measurement methods provide principal strains of formed sheet metal parts and compute the thickness [1]. These systems are very useful for forming processes and making tools etc. in industry in the last years and are used widely part design, quality control and die sinking etc. More precise and quick measurements can be carried out with the aid of an automated strain measurement method.

The purpose of this study is to compare the numerical and experimental results in hemispherical punch stretching test. The comparison of measured and simulated strain and thickness reduction distributions were evaluated. In addition, dimensional comparison was performed by measuring dimensions of parts with using optical 3D scanning device.

EXPERIMENTAL STUDY

In order to validate the FEM results, formed parts by hemispherical punch were used. AA 5754-O sheet material parts which have 1 mm thickness, 50 mm and 200 mm widths were clamped between circular die rings and were deformed by a hemispherical punch until the first fracture observed. The 50 mm width was used for deep drawing condition and the 200 mm width was used for stretching condition **(FIGURE 1)**. The punch diameter is 100 mm. Approximately 260 kN blank holder force was applied in order to not allow to drawing in the specimens during the test. Three repeated tests were conducted for both specimens. Before the test, samples were marked with 2.5x2.5 mm line grid patterns by using serigraphy method to measure the deformations after the test. The grids were exhibited an accuracy of 0.28 % and repeatability of 0.8 % [6]. These values are below the 1% specified in ASTM E2218-02 Standards.

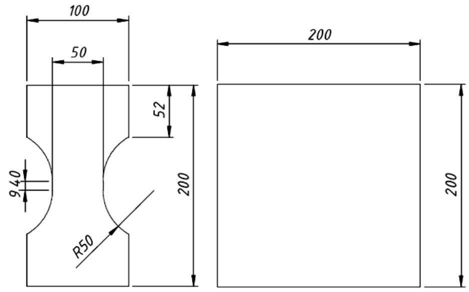


FIGURE 1. The specimens used measuring strains on them.

Measurement of Strain Distribution

Commercially available automated strain analysis and measurement environment system (ASAME software) was used to experimentally measure the strains in the parts in order to verify the FEM results. A target cube with the 25mm was placed the area to be measured and photographs were then taken from two different viewpoints using the 12 MP resolution digital SLR camera. The photographs were then processed using the ASAME software [5]. Accuracy of ASAME Target Model is denoted as 1.5 % [3]. All measurements were made with $\pm$0.38 % accuracy, 0.26 % repeatability and with confidence of 95% [6]. In order to compare with FEM results, two sections that pass from centre of the dome were drawn radially on the each part. The strains at the sections were compared with FEM results.

3D Measurement of Parts

The parts geometry and the thickness distribution were compared with FEM results. Breuckmann Opto Top-HE 3D Optical Scanning System [7] was used for measuring of formed parts' geometry. Firstly, the parts were painted with white dust spray. The reference stickers, as called index markers, were labeled onto the parts before the painting as shown in **FIGURE 2**. Usually, these types of parts are scanned the top and the bottom surface of the specimen separately and then superposed the surfaces. In this case, some errors occur. Whole scanning of the parts removes the need of the superposition of the surfaces. However, it is required that the optical device perceives the least three marker at the same time during the scanning. Therefore, it is also required that scanned two surfaces must include common markers. Because the parts are sheet metal, it is impossible that such an angle is obtained on the parts. In order to obtain this requirement, a rectangular parallelepiped reference part was gummed on the corner of the part **(FIGURE 3)** and was marked with marker before the painting. The parts were scanned and the point data were obtained for the top and bottom surfaces. Extra superposition was not needed. Now we can investigate dimensions of the parts.

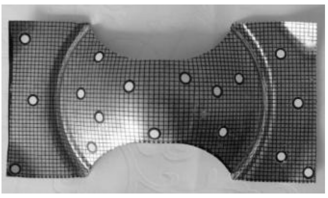


FIGURE 2. The part (having 50 mm width) with marker

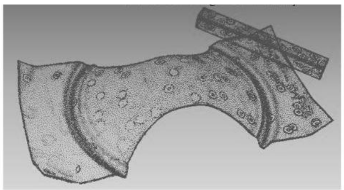


FIGURE 3. The rectangular parallelepiped was gummed on the corner of the sheet part

NUMERICAL STUDY

The processes of stretching of two geometries were modeled with ABAQUS using shell and axisymmetric elements. The FEM models of the tools were designed. Due to the stretching process, the blank holder was not modeled. Instead of this, ancestry boundary condition was applied at the areas of corresponding to draw bead region. The specimens were modeled for shell elements. For the specimen, 4 node shell elements were used and the two dimensions of the elements were tried to close each other. For axisymmetric modeling of 200 mm width, the specimen was divided to 50 elements having the same dimensions. Due to the symmetry conditions, a quarter of the specimens were used for shell elements in order to reduce the total number of nodes and elements and therefore, to accelerate the calculations.

FIGURE 4 shows the FEM models of the stretching using shell elements. The die and the punch were modeled as analytical rigid parts. The material properties of AA 5754 were obtained from tensile test then elastic properties and plastic yield curve were defined to the program. The material was assumed to isotropic.

Surface integrations were defined as surface to surface contact. The friction coefficient μ between the tools and the specimen were determined by trial and error. The μ which gives the best well suited strain distribution was used in the simulation. The μ was chosen between 0.27 and 0.36 in the simulations. After the simulation has been completed, the major strain distribution of the part was obtained as shown in **FIGURE 5**. The maximum strain values were developed at the same regions of the real

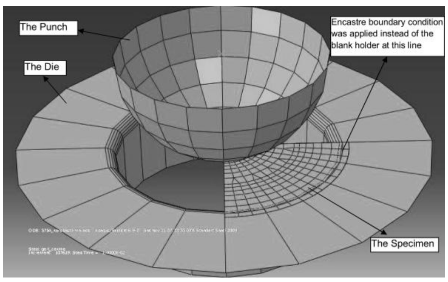


FIGURE 4. FEM model of the stretching process

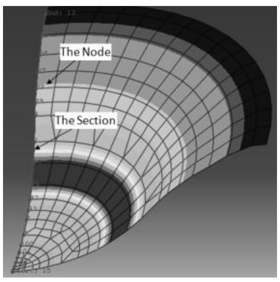


FIGURE 5. Major strain distribution and the comparison section of the 200 mm width

test specimens. In order to compare FEM results with test results a section which starts at the pole of the dome and finishes at the end of the specimen was used **(FIGURE 5)**. The 3D coordinates, the thicknesses and the major strain values of these nodes were compared corresponding sections on the real test specimen.

RESULTS AND DISCUSSION

Comparison of the strain distributions of the measured and simulated the 200 mm width part are seen in **FIGURE 6**. Every line indicates a major strain distribution on a different section. For a part, the strains were obtained from four different sections

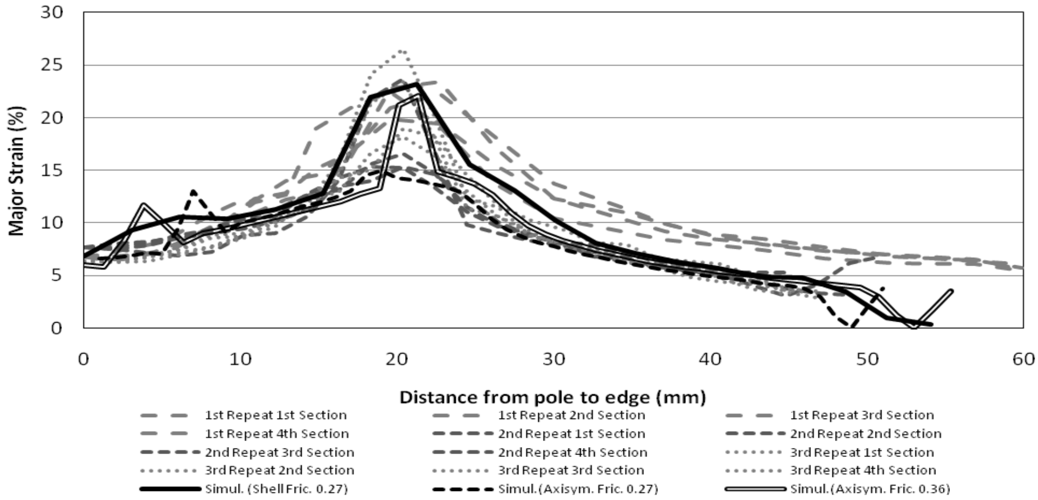


FIGURE 6. Comparison of the strain distributions of the measured and simulated 200 mm width part

which pass from pole to the edge of the part. Measurements were obtained from three repeated tests. As seen in the figure, the simulation result for shell elements with μ of 0.27 and for axisymmetric elements with friction coefficient of 0.36 were well correlated with the test results. As a result both of the element types may be convenient for the simulations.

The same situation was obtained for the 50 mm width specimens as seen in **FIGURE 7**. For these specimens only one section could be obtained which passes from pole to the edge as longitudinally. Obtaining so compatible simulation results were accomplished by arranging the mesh structure and the μ between the sheet and the punch. These two parameters were arranged to obtain the best compatible FEM results. For example for 200 mm width specimen when the μ was selected 0.27, the results shown in the figure were obtained. But in order to obtain compatible results with the tests, the μ was selected 0.36 for 50 mm width specimen. When the same friction

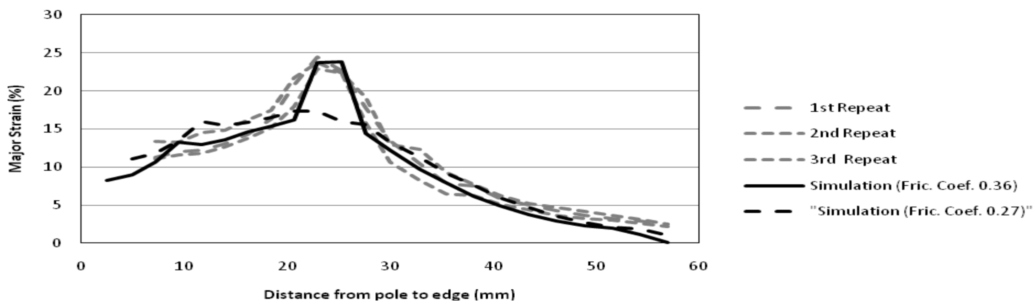
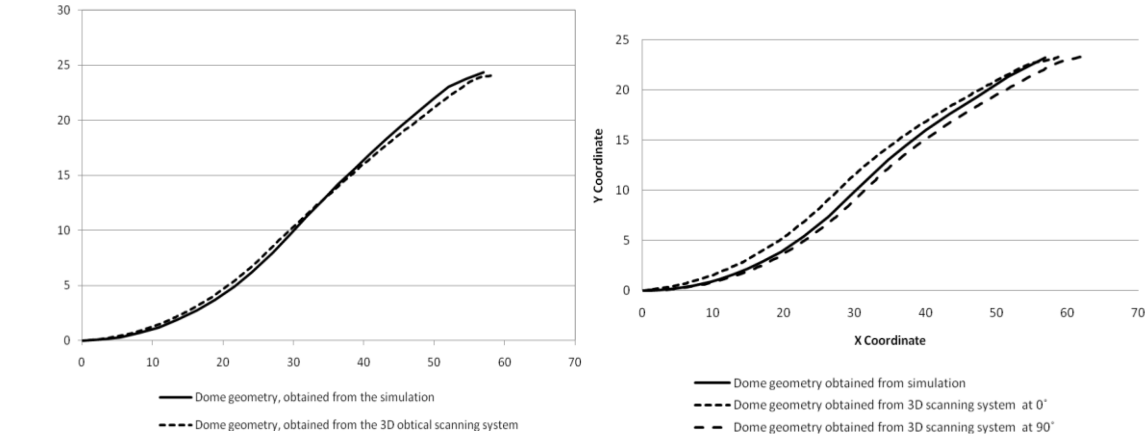


FIGURE 7. Comparison of the strain distributions of the measured and simulated 50 mm width part



a) b)

FIGURE 8. Geometric comparison of a) 50 mm and b) 200 mm width parts obtained with simulation and 3D scanning

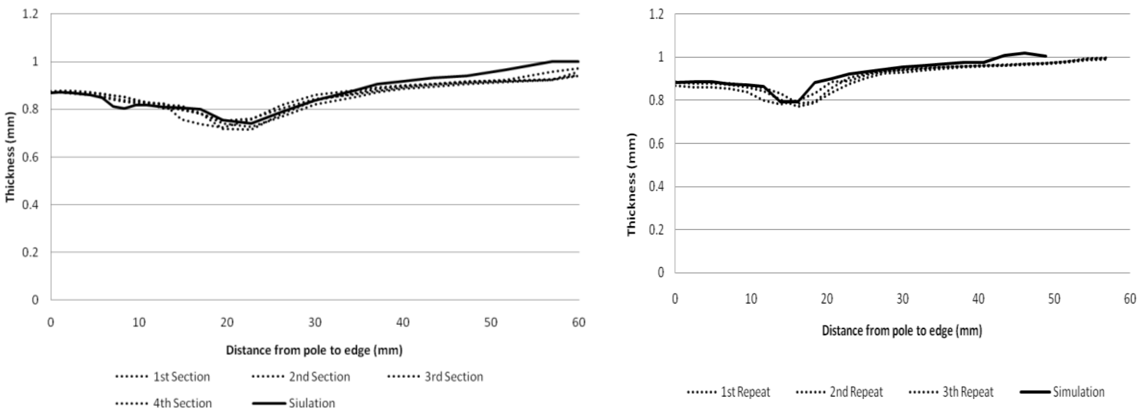
coefficient (0.27) was selected for this specimen there are some differences between the test and the simulation results **(FIGURE 8)**. This difference was found approximately 27% for the maximum strain value region and when the whole area was considered, this difference becomes approximately 6%. Besides, the maximum major strain region changes with the μ. As the μ reduces, the maximum strain region get closer to the pole. It is clear that using of the axisymmetric elements could not be used because of the geometry. Consequently the major strain distribution of both parts could be predicted well with FEM simulations.

Geometrical validation of the simulation was conducted by comparison of the measured coordinates of the formed parts by 3D scanning system and the simulations along the mentioned sections above. The results of geometrical comparison are given graphically in **FIGURE 9** and numerically in **TABLE 1**. Consequently, predicted geometry by simulation of the two parts is correlated well with the test results. Usage of the different element types in the simulations did not affect the geometrical results.

TABLE 1. Geometric comparison of the parts and simulations

Distance	200 mm specimen		50 mm specimen
	Section 1	Section 2	
Maximum	1.42 mm	1.24 mm	0.69 mm
Average	0.77 mm	0.54 mm	0.33 mm

Thickness comparison was conducted between the simulation results and the computed thicknesses by ASAME along the mentioned sections above **(FIGURE 9)**. Good results were obtained by simulation for most of the section except edges of the parts. The thicknesses vary 2% as average and 6.7% as maximum in accordance with



a) b)

FIGURE 9. Geometric comparison of a) 50 mm and b) 200 mm obtained with simulation and 3D scanning

the test results. The difference becomes greater at the edges. Because in the tests the draw bead allows the material’s thinning, but in the simulations the edges are considered as ancestry.

CONCLUSIONS

In this study a comparative work has been done with the purpose of showing accuracy of the FEM results. For this a dome stretching test had been done and surface strains, sheet thicknesses and surface geometry were measured and compared with the results of the same forming process’s simulation.

The following results have been concluded.

- Shell and axisymmetric elements can be used for the simulation of the stretching processes.
- The differences between the test and the simulation are averagely 2% for thicknesses and 6% for major strains. The difference of the geometrical coordinate changes between 0.33-0.77 mm. Consequently the FEM results may be accepted as compatible with the test results.
- The most important parameter which affects the simulation results of sheet metal forming process is mesh structure and the friction between the sheet and the tools. The simulation results differs approximately 27% for a particular μ when the geometry varies.

For future works, firstly the μ must be determined by friction test. Then in order to obtain compatible simulations, other parameters and the mesh structure should be determined.

ACKNOWLEDGMENT

This work is supported by The Scientific and Technological Research Council of Turkey (TUBITAK). Project number: 108M516. Project Title: “Adding drawbead to the blank holder for enhancing formability of aluminum alloy sheets with hydroforming process”. This work is also supported by the BAP office of Selcuk University. TUBITAK and BAP support is profoundly acknowledged.

REFERENCES

1. Schneider M. et al, 2008, Validation and Optimization of Numerical Simulations by Optical Measurements of Tools and Parts, International Deep Drawing Research Group IDDRG 2008 International Conference, 16-18 June 2008, Olofström, Sweden.
2. Andersson A., 2004, Comparison of sheet-metal-forming simulation and try-out tools in the design of a forming tool, J. Eng. Design, Vol. 15, No. 6, pp. 551–561.
3. ASAME (Automated Strain Measurement Analysis and Measurement Environment) Manual.
4. www.gom.com
5. Vogel J.H. and Lee D. , 1990, The Automated Measurement of Strains From Three-Dimensional Deformed Surfaces, Journal of The Minerals, Metals & Materials Society, Vol. 42, No. 2, pp. 8-13.
6. Ozturk F. et al, 2009, Grid Marking and Measurement Methods for Sheet Metal Formability, The 5th Inter. Conf. and Exhi. on Design and Product. of Machines and Dies/Molds, Turkey, June 18-21.
7. http://www.accurexmeasure.com