Journal for Technology of Plasticity, Vol. 36 (2011), Number 2

DOI: 10.2478/v10211-011-0009-9

NUMERICAL AND EXPERIMENTAL COMPARISON OF HEMISPHERICAL PUNCH STRETCHING TEST

Mevlut Turkoz[1], Osman Yigit[2], Murat Dilmec[3],
Huseyin Selcuk Halkaci[3], Behcet Daghan[3]

[1]Department of Seydisehir Mechanical Engineering, Selcuk University, Konya, Turkey
[2]Karabuk University, Department of Mechanical Engineering, Karabuk, Turkey
[3]Selcuk University, Department of Mechanical Engineering, Konya, Turkey

ABSTRACT

During the recent years, a huge amount of 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 research, a comparative study was performed between experimentally and numerically obtained strains, part geometries and sheet thicknesses. Different sheet metal specimens were clamped between circular die rings and deformed by a hemispherical punch. Before the sheets had been formed, they 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 to this, 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*

1. 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.

2. EXPERIMENTAL STUDY

In order to validate the FEM results, formed parts by hemispherical punch were used. In the study, 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 specimen that has 50 mm width was used for deep drawing condition and the specimen that has 200 mm width was used for stretching condition (Fig. 1). The diameter of punch is 100 mm. Approximately 260 kN blank holder force was applied in order to not allow to drawing in the specimens during the test. The specimens were formed at 25 mm/min punch speed. Three repeated tests were conducted for both specimens. Before the test, samples were marked with line grid patterns having 2.5x2.5 mm dimensions 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.

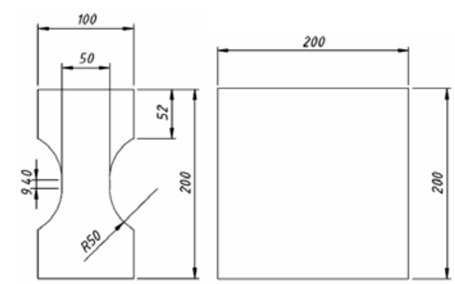


Fig. 1 **-** *The specimens used measuring strains on them.*

2.1 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. ASAME measures both surface strains and surface geometry of a large area by the aid of two images which have a geometric relationship to one another. In order to determine the strains in the parts, initially photographs of all areas of the parts by dividing several areas were taken. For example, the part having 200 mm width was divided thirteen areas. Before the photographs are taken, a target cube with the 25mm was placed the area to be measured, and photographs were then taken from two different viewpoints using an 12 MP resolution digital SLR camera. Each photograph must include the target cube and the measurement area. The photographs were then

processed using the ASAME software in order to obtain the surface geometry and compute major and minor strains [5]. Accuracy of ASAME Target Model is denoted as 1.5 % [3]. All measurements were made with ± 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 and that are orthogonal each other were drawn radially on the each part. The strains at the sections were compared with FEM results. Therefore the areas were evaluated where near of the centre section.

2.2 3D Measurement of Parts

The parts geometry and the thickness distribution were compared with FEM results. Breuckmann Opto Top-HE 3D Optical Scanning System was used for measuring of formed parts' geometry in the study. This measurement system provides a dense cloud of three dimensional coordinates for the surface of the part being measured [7]. In this measurement system, 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 Fig.2. Usually, these types of sheet 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 reference part which has a geometry rectangular parallelepiped was gummed on the corner of the sheet part **(Fig.3)**, marked with marker before the painting. The parts were scanned and the point cloud data were obtained for the specimens' top and bottom surfaces. Extra superposition process was not needed. Now we can investigate dimensions of the parts.

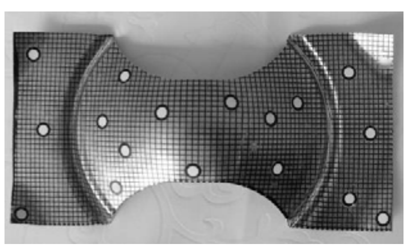


Fig. - 2 The part (having 50 mm width) with marker

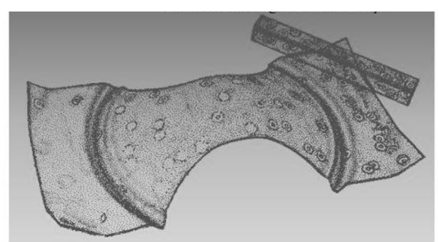


Fig. 3 - The rectangular parallelepiped was gummed on the corner of the sheet part

3. NUMERICAL STUDY

The processes of stretching of two different geometries were modeled with the FEM simulation program ABAQUS using shell and axisymmetric elements. The FEM models of the tools were designed to match the geometrical dimensions of the die and punch. Due to the stretching process, the blank holder was not modeled. Instead of the blank holder, ancestry boundary condition was applied at the areas of corresponding to draw bead region. The specimens having two different widths were modeled with the same geometric dimensions of stretching process 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.

Figure4 shows the FEM models of the stretching process 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 between the tools and the specimen were determined by trial and error. The friction coefficient which gives the best well suited strain distribution was used in the simulation. The friction coefficient 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 Figure5. The maximum strain values were developed at the same regions of the real test specimens and it was shown as red color in the figure. 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 (Fig.5). The 3D coordinates, the thicknesses and the major strain values of these nodes were compared corresponding sections on the real test specimen.

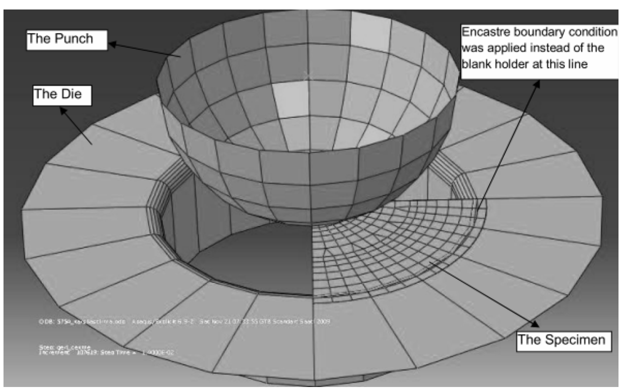


Fig.4 - FEM model of the stretching process

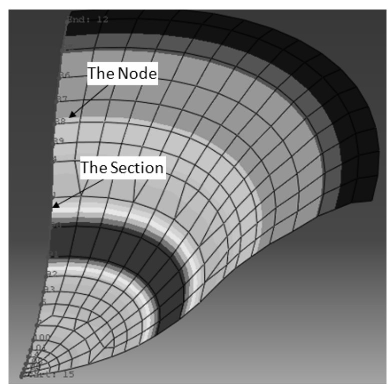


Fig.5 - Major strain distribution and the comparison section of the 200 mm width

4. 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 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 friction coefficient 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.

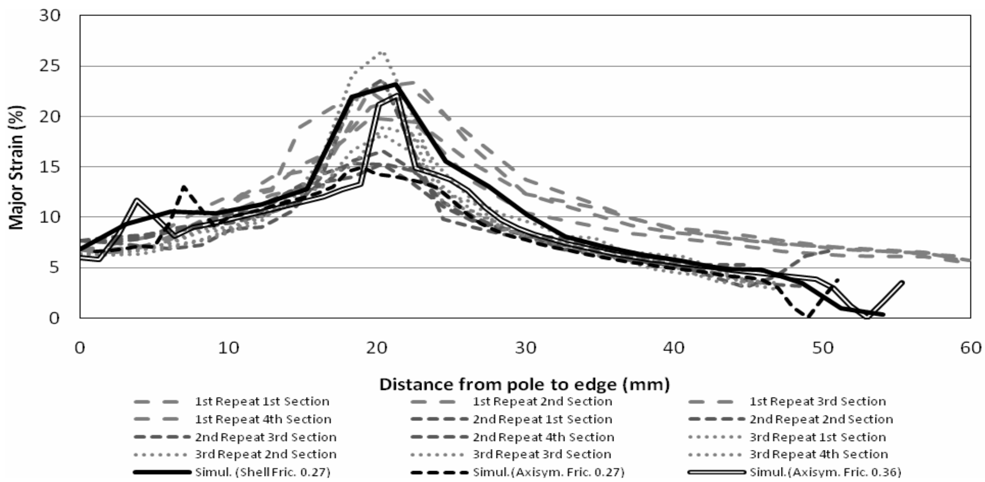


Fig.6 - Comparison of the strain distributions of the measured and simulated 200 mm width part

The same situation was obtained for the 50 mm width specimens as seen in Fig.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 friction coefficient 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 friction coefficient was selected 0.27, the results shown in the figure were obtained. But in order to obtain compatible results with the tests, the friction coefficient was selected 0.36 for 50 mm width specimen. When the same friction coefficient (0.27) was selected for this specimen there are some differences between the test and the simulation results (Fig.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 friction coefficient. As the friction coefficient 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 the two different geometrical parts could be predicted well with FEM simulations.

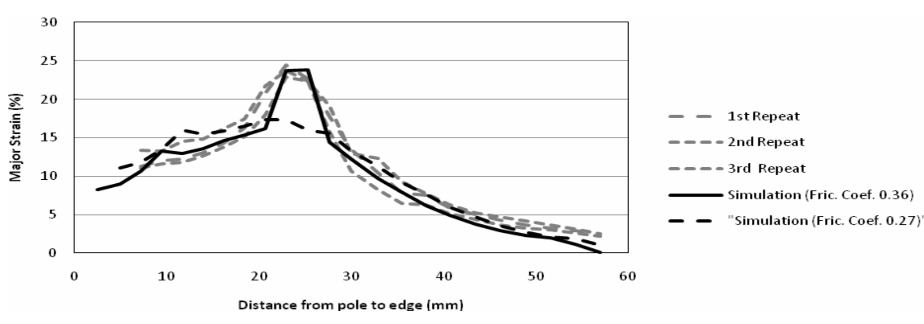
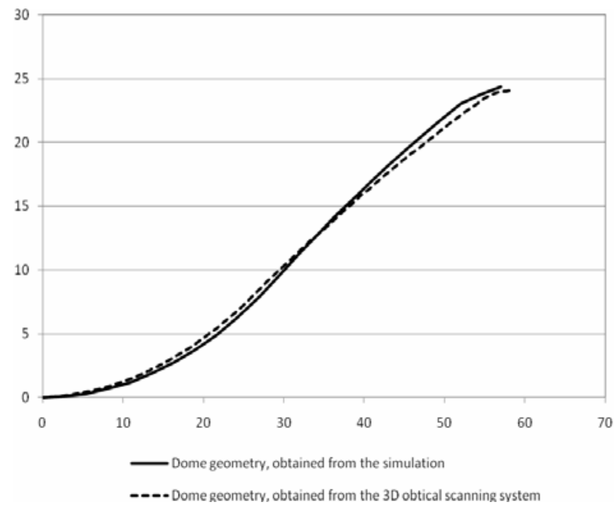
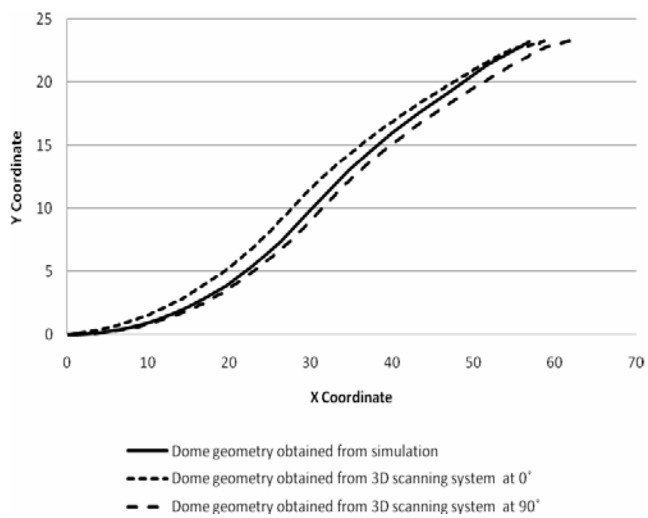


Fig. 7 - Comparison of the strain distributions of the measured and simulated 50 mm width par



a)

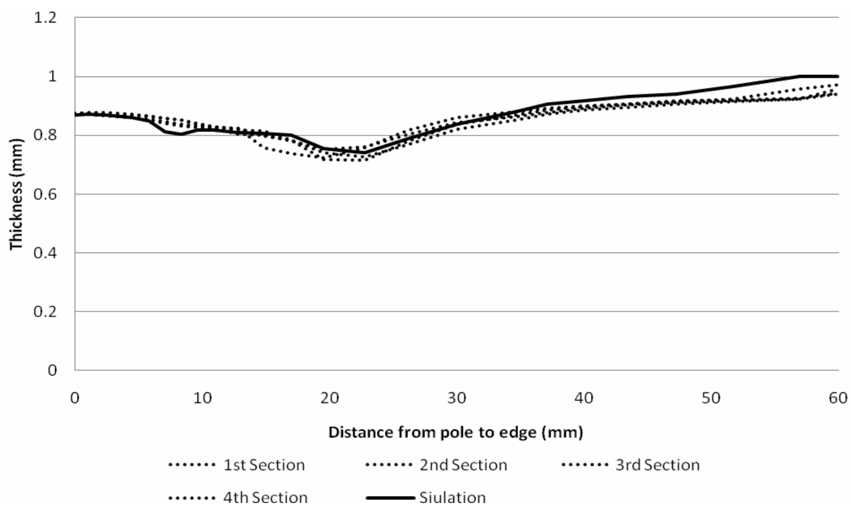


b)

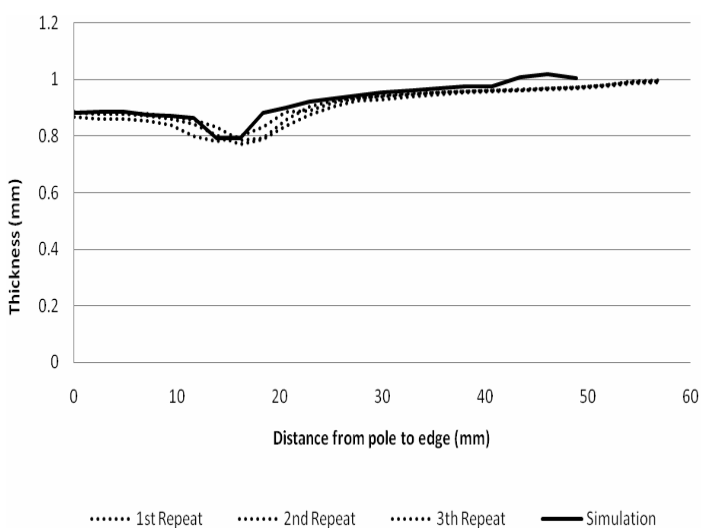
Fig.8 - Geometrical comparison of a) 50 mm and b) 200 mm width parts obtained with simulation and 3D scanning system

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 in Fig.9 As seen in the figure predicted geometry by simulation of the two parts are correlated well with the test results. Usage of the different element types in the simulations did not affect the geometrical results.

Thickness comparison was conducted between the simulation results and the computed thicknesses by ASAME along the mentioned sections above. The comparison was shown in Fig. 10. Good results were obtained by simulation for most of the section except edges of the parts. The thicknesses vary approximately 5% at the edges in accordance with the test results. Because in the tests the draw bead allows the material's thinning, but in the simulations the edges are considered as encastre.



a)



b)

Fig.9 - Thickness comparison of a) 200 mm b) 50 mm width parts obtained with simulation and ASAME

5. 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.
- Major strains, thicknesses and geometrical dimensions obtained from the tests are compatible with the FEM simulations.
- 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 friction coefficient when the geometry varies.

For future works, firstly the friction coefficient 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) Instruction 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 Int. Conf. and Exh. on Design and Product. of Machines and Dies/Molds, Turkey.

[7] http://www.accurexmeasure.com

NUMERIČKA I EKPERIMENTALNA KOMPARACIJA TESTA ISTEZANJA HEMISFERIČNIM ŽIGOM

Mevlut Turkoz[1,a], Osman Yigit[2,b], Murat Dilmec[3,c],
Huseyin Selcuk Halkaci[3,d], Behcet Daghan[3,e]

[1]Department of Seydisehir Mechanical Engineering, Selcuk University, Konya, Turkey
[2]Karabuk University, Department of Mechanical Engineering, Karabuk, Turkey
[3]Selcuk University, Department of Mechanical Engineering, Konya, Turkey

REZIME

Tokom poslednjih godina vršena su mnogobrojna istraživanja koja su posvećena numeričkim metodama simulacije obrade lima. Cilj ovih istraživanja je smanjenje perioda eksperimentalnog testiranja. U ovom istraživanju izvršena je komparativna studija između eksperimentalno i numerički dobijenih deformacija, geometrija delova i debljina lima. Različiti uzorci od lima su spojeni između cirkularnih prstenova matrica i deformisani hemisferičnim žigom. Pre nego što su limovi deformisani, markirani su sa mrežom koja omogućuje merenje deformacije nakon eksperimenta. Merenje deformacija i preračunavanje debljine je izvršeno pomoću Automated Strain Analysis System-a (ASAME). Procene deformacija su vršene pomoću ABAQUS-a. Kako dodatak, izvršena je i komparacija geometrije merenjem dimenzija uzoraka pomoću 3D optičkog uređaja za skeniranje.

Ključne reči: *Obrada lima, Metod konačnih elemenata, ABAQUS, ASAME, 3D skeniranje*