Journal of Mechanical Science and Technology 28 (1) (2014) 237~243

www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-013-0959-0

Finite element simulation of plate or sheet metal forming processes using tetrahedral MINI-elements[†]

Wanjin Chung[1], Bongsu Kim[2] , Seongwon Lee[3], Hoyeun Ryu[3] and Mansoo Joun[2,*]

[1]*Dept. of Mechanical System Design Engineering, Seoul National University of Science and Technology, Seoul, Korea*
[2]*School of Mechanical Engineering / Engineering Research Institute, Gyeongsang National University, Jinju, Korea*
[3]*Korea Institute of Industrial Technology, Jinju, Korea*

(Manuscript Received February 24, 2013; Revised July 25, 2013; Accepted August 13, 2013)

Abstract

In this paper, finite element prediction of sheet metal forming process is investigated using solid elements. A three-dimensional rigid-viscoplastic finite element method with linear tetrahedral MINI-elements is employed. This technique has traditionally been used for bulk metal forming simulations. The solid element approach with remeshing capability is applied to simulating a plate metal forming process to reveal its possible problems. The similar approach is also applied to a typical sheet forming process. Both single- and double-layer finite element mesh systems are studied, with particular attention to their effect on the deformed shape of the workpiece and thickness variation of a cold sheet forming process. The procedure is applied to the well-known problem of the NUMISHEET93 international benchmark. The resulting predictions are compared with experimental observations found in the literature, and good agreement is noted.

Keywords: Plate metal forming or plate forging; Rigid-plastic finite element method; Sheet metal forming; Tetrahedral MINI-element

---

1. Introduction

Metal forming simulations for sheet or plate metal products have become essential to meet the needs of industries in applying new materials and reducing development cost and time. The most recent sheet metal forming simulators are capable of accurately handling various material models, including the popular elastoplastic model, as well as complex workpiece, die, and tool geometries, various working conditions, and consecutive forming processes [1].

Simulations of innovative sheet or plate metal forming processes, including hydroforming and hot sheet metal forming processes, are currently being attempted. Springback analysis provides process design engineers with much valuable information for improving dies or tools, and enhancing the tolerance of products [2].

However, the shell elements that lead to numerical simplicity and computational efficiency are subject to distinct limitations when applied to plastic deformation in the thickness direction. To make matters worse, traditional shell elements are very weak when it comes to simulating thick sheet or plate metal forming processes with small corner radii [3, 4]. For these reasons, shell elements are not appropriate for sheet or plate metal forming simulations to obtain detailed predictions of plastic deformation in the thickness direction. Sheet metal forming processes designed to produce an abrupt thickness change via the forging technique (which are not readily treated as traditional sheet metal forming processes) are currently being applied in the automotive and electronic industries to meet the need for both structural rigidity and light weight. A plate metal forming process can be considered as a special type of sheet metal forming process (i.e., a special forging technology married to the sheet metal forming process). Up to now, process design in these areas has been empirical in nature, and hence an appropriate technique is necessary for modeling the processes mathematically.

It is noteworthy that structural parts formed from sheet metal are especially weak around bent corners, and thus detailed information on plastic deformation around corners is a major interest of process design engineers. As was previously mentioned, traditional sheet metal forming simulation technologies are distinctly inadequate to satisfy these interests. Solid elements can be used to remedy this issue (in other words, to more accurately describe plastic deformation in the thickness direction and around corners). Thus far, solid elements or solid-shell elements have been used to account for plastic deformation in the thickness direction. Xu et al. [5] and Parente et al. [6] developed a solid-shell element model in which multiple integration points along the thickness are used to

---

[*] Corresponding author. Tel.: +82 772 1624, Fax.: +82 772 1624
E-mail address: msjoun@gnu.ac.kr

[†] Recommended by Editor Yeon June Kang

describe a bending effect, while a selective reduced integration technique is employed in the plane. However, solid-shell elements can be used only for a sheet metal forming process with a gradual thickness change. Recently, a number of researchers have studied tetrahedral or hexahedral elements for modeling three-dimensional sheet or plate metal forming processes. Menezes et al. [7] utilized eight-node brick elements to analyze square-cup drawing. Lee et al. [8] applied linear tetrahedral elements to a three-dimensional thick sheet metal forming process, and demonstrated the feasibility of such a technique.

When solid elements (including tetrahedral MINI-elements [9, 10]) are employed, intrinsic mesh generation problems prevent simulation technology researchers from developing application-oriented software. Due to the geometric characteristics of sheets and plates, the use of inherently bad elements with high aspect ratios is unavoidable when the number of elements is constrained. Similarly, multi-layer mesh systems should be utilized at the expense of both mesh quality (due to increased aspect ratio) and computational efficiency (due to the increased number of finite elements and nodes) to control the mesh density in the thickness direction when conventional solid elements are employed.

In this paper, a sold element approach to plate or sheet metal forming simulation is investigated. A plate metal forming process is studied to reveal its numerical characteristics and applicability and the effect of the number of layers in a multi-layer finite element mesh system on solutions of sheet metal forming simulations is investigated.

2. Simulation of a plate metal forming process

Fig. 1 shows a plate metal forming or plate forging process to be simulated. Thickness and area of the workpiece are 7.5 mm and 550.0 by 550.0 mm, respectively. One of the geometrical features of this process is its thickness ratio to the area, which is much greater than that of a common sheet metal forming process. One of major concerns in process design of this process is the thickness change around the corners. A relatively greater thickness ratio of the workpiece compared to its area and the major interest of thickness distribution around die corners make a good environment for application of bulk or solid finite elements to its simulation.

It is assumed that the strength coefficient and strain-hardening exponent of the material are 730 MPa and 0.22, respectively and that the interface between die and workpiece obeys law of Coulomb friction with its coefficient of 0.05 assumed. A rigid-plastic finite element method with intelligent remeshing capability [10, 11] that has been optimized for precision simulation of bulk metal forming processes is employed to reflect or describe local change of curvature or deformation and workpiece-die interface in detail. It is because artificial or numerical change of workpiece surface leads inevitably to direct change of the thickness of workpiece that penetration of the nodes on free surfaces into dies should be severely controlled in updating the workpiece to predict the thickness with higher accuracy and that the old surface of workpiece should be accurately mapped into the new surface during remeshing.

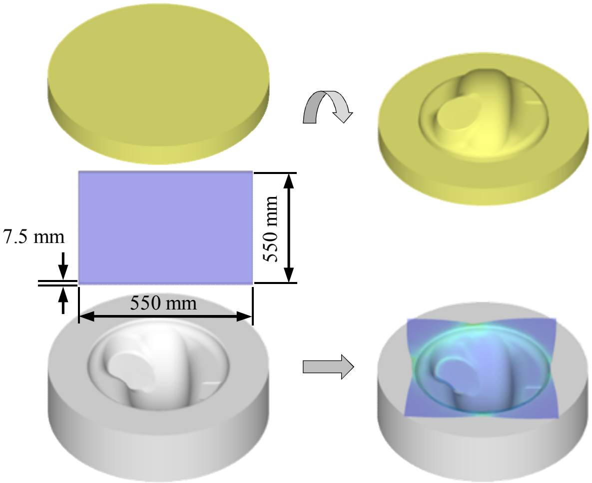


Fig. 1. Process design of a plate metal forming process.

After 350 solutions steps and 8 remeshings, the predictions were obtained. First of all, it was checked that the volume loss was less than 0.95% even though no artificial volume loss compensation scheme was utilized, implying that the penetration of workpiece into dies was properly controlled during simulation and surface geometry was well maintained or transferred during remeshing, that is, the predictions are reliable. Fig. 2 shows the predictions of history of deformation and Fig. 3 shows the distribution of effective strain. The number of tetrahedral elements is between 39001 at the initial solution step and 54107 at the final solution step.

Fig. 4 shows the distribution of plate thickness which ranges from 5.33 to 8.34 mm and varies smoothly, implying also that the predictions are reliable. However, the number of elements in the thickness direction is so small due to the extremely high aspect ratio and thus the finite element mesh system is single-layered in some regions. This fact can deteriorate the solution accuracy especially when compressive and tensile stresses are exerted in the thickness direction at the same time. The bad mesh quality in the thickness direction is inevitable when a common function of remeshing developed or optimized for bulk metal forming processes is employed. It is thus concluded that a special function of multi-layer mesh generation technique should be supported for precision simulation of the sheet or plate metal forming processes with solid elements because the stresses on opposite sides of the workpiece generally have opposite states in sheet or plate metal forming (compression on one side and tension on the other side), especially in the region where bending deformation is dominant.

3. Simulation of a sheet metal forming process

When solid elements are used to carry out a finite element

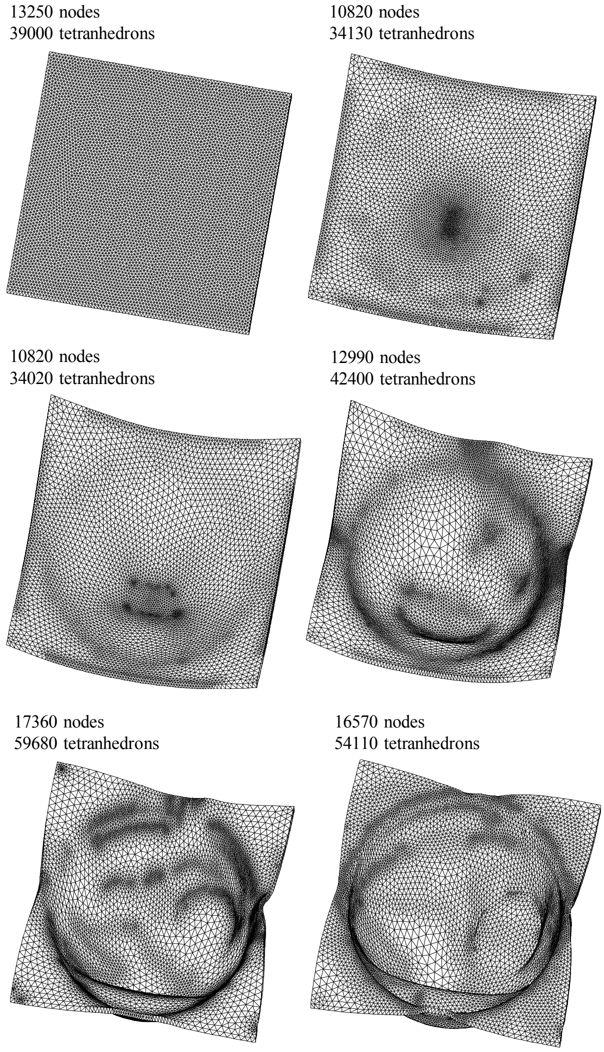


Fig. 2. History of deformation.

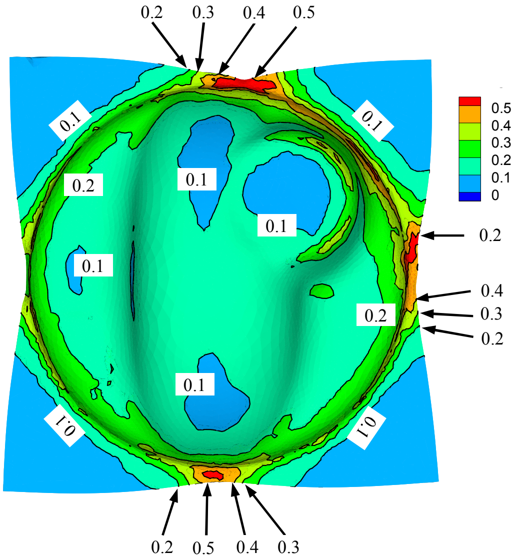


Fig. 3. Effective strain.

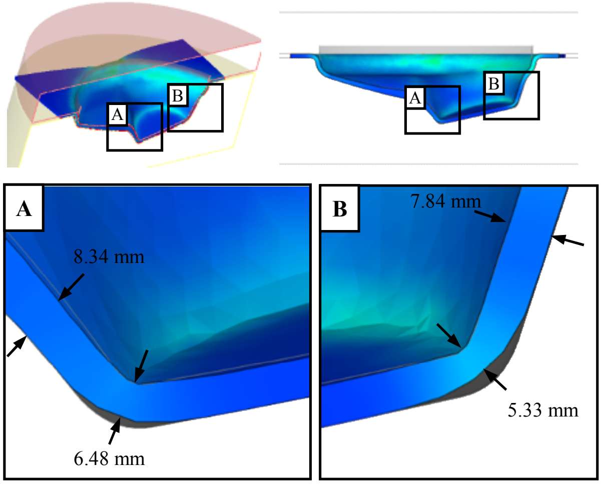


Fig. 4. Variation of thickness at a cross-section.

simulation of a sheet or plate metal forming process, it is essential to construct a finite element mesh system with the proper number of elements in the thickness direction, or to develop special finite elements to reflect the bending deformation of the workpiece. It is known that special solid elements, including solid-shell elements [5, 6], can be helpful for dealing with this problem. If traditional solid elements are employed to simulate a sheet or plate metal forming process, the effect of the number of integration points in the thickness direction on finite element solutions should first be investigated in detail. When linear solid elements are utilized, the number of integration points in the thickness direction can be controlled by the number of layers in a multi-layer finite element mesh system. However, there is a distinct limitation on how much the number of layers can be increased, due to the corresponding increase in the number of degrees of freedom, which directly affects the computational time. Also, as the number of layers in a multi-layer finite element mesh system increases, the aspect ratio increases. Therefore, it is necessary to consider the appropriate number of finite element mesh layers.

In this paper, a sheet metal forming process was first simulated by single- and double-layer mesh systems, and the predictions were compared to quantitatively determine the effect of the number of layers on the finite element solutions. The rigid-plastic finite element method with tetrahedral MINI-elements [10, 11] were also adopted for the analysis.

The sheet metal forming process to be simulated is shown in Fig. 5 [12]. It is the square-cup deep drawing problem associated with the NUMISHEET93 international benchmark. The blank is a 150 × 150 × 0.78 mm square sheet. The flow stress of the mild steel used is $\bar{\sigma} = 566(0.007 + \bar{\varepsilon})^{0.259}$ MPa, and the yield strength is 167.0 MPa. A binder force of 19.6 kN is exerted, and will be represented by the equivalent frictional force with the binder fixed with respect to the lower die. The punch velocity was assumed unity in that it does not affect the solution under the rigid-plastic assumption.

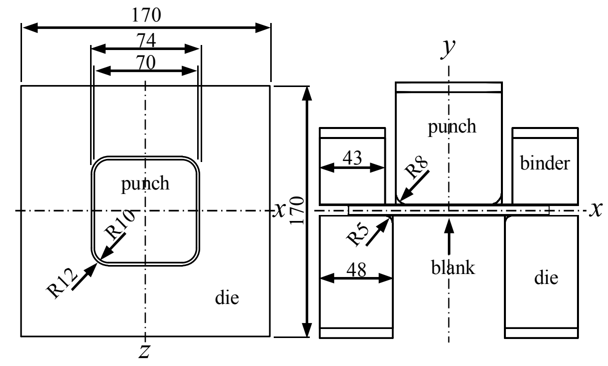


Fig. 5. Definition of the square-cup drawing process.

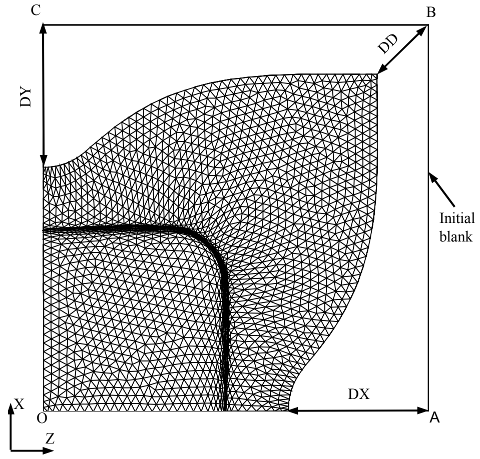


Fig. 6. Definition of the measuring distances.

Fig. 6 defines the measuring distances DX, DD, and DY along OA, OB, and OC, which determine the deformed geometries for comparing the predictions of the present technique with other predictions [5] obtained using solid-shell element or shell element and the experiments found in the Ref. [12].

An analytical model was constructed as follows. To approximately represent the binder force in the model, the law of constant shear friction was adopted, with a shear factor of 0.02 and the distance between the binder and the lower die fixed as the initial thickness of the workpiece, augmented by a small tolerance of 0.2 mm (i.e., 0.98 mm). To obtain the mathematical models for the tools, the binder, punch and die were discretized into 20, 736 and 872 triangular patches, respectively, considering the requirement for geometric tolerance, as shown in Fig. 7.

Fig. 8 shows the finite element mesh systems used to simulate the test example. Note that only one fourth of the process was required for the analysis domain, owing to its symmetry. The numbers of tetrahedral elements and nodes in the single-layer finite element mesh system were 13830 and 4790, respectively. The double-layer finite element mesh system was constructed by mirroring the single-layer system.

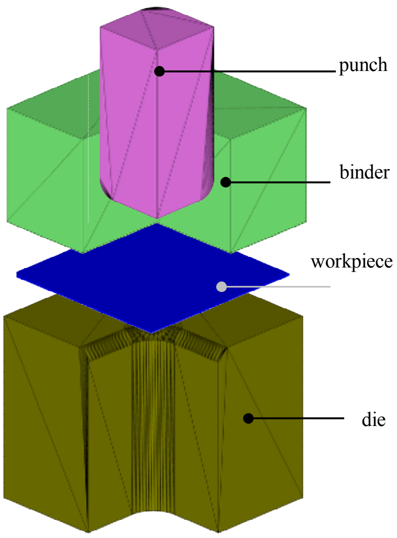
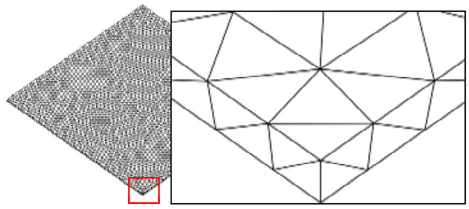
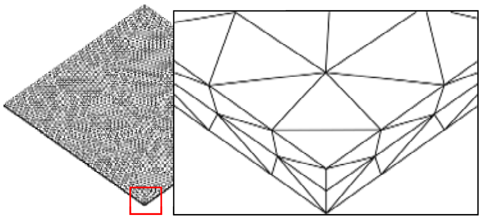


Fig. 7. Three-dimensional view of the test process.



(a) Single-layer



(b) Double-layer

Fig. 8. Finite element mesh systems for the workpiece.

The process simulation was carried out for two different finite element mesh systems under the same conditions, using the rigid-plastic finite element method [11, 13]. Remeshing during the simulation was excluded to avoid numerical smoothing of geometrical dimensions and state variables.

The solutions were obtained after 1000 steps. Figs. 9 and 10 show the predictions of the final deformed shape and effective strain for the single- and double-layer finite element mesh systems, respectively. The total numerical volume loss was less than 0.4%, indicating proper control of the numerical uncertainties concerned with the contact algorithm and workpiece penetration into the die or die penetration into the workpiece (which may be one of the inherently bad characteristics of solid elements for sheet metal forming simulations).

The measured distances DX, DY, and DD at a punch stroke of 40 mm were compared with the experimental results and

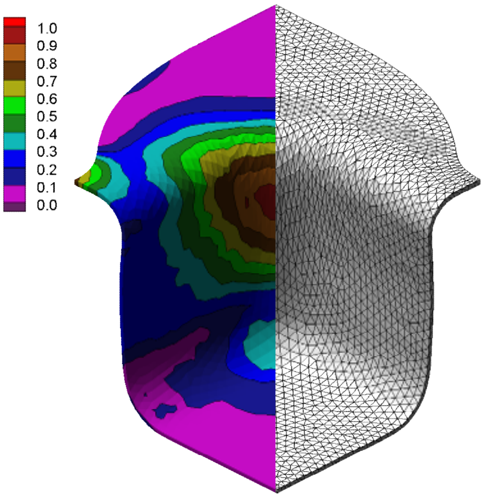


Fig. 9. Effective strain and final deformed shape (single-layer).

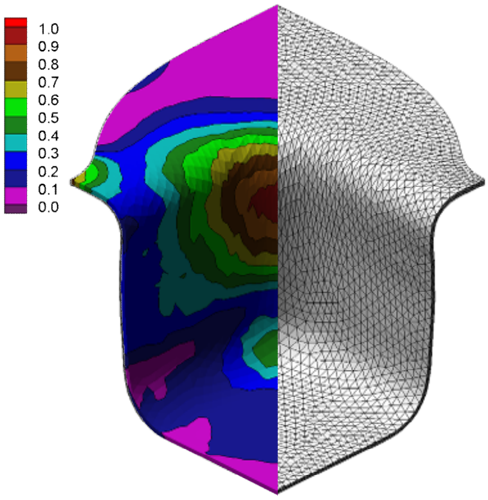


Fig. 10. Effective strain and final deformed shape (double-layer).

Table 1. Comparison of the predictions with the experiments and other predictions.

	DX, DY(mm)	DD(mm)
Experiment results [12]	27.95	15.36
Tetrahedral, single-layer	27.61	13.85
Tetrahedral, double-layer	26.50	13.15
Solid-shell(Xu. et al.) [5]	27.17	14.79
LS-DYNA3D (shell) [12]	30.03	16.43

Table 2. Mesh size effect.

	DX, DY(mm)	DD(mm)
Tetrahedral, single-layer (13830 elements)	27.61	13.85
Tetrahedral, single-layer (28190 elements)	27.19	13.69

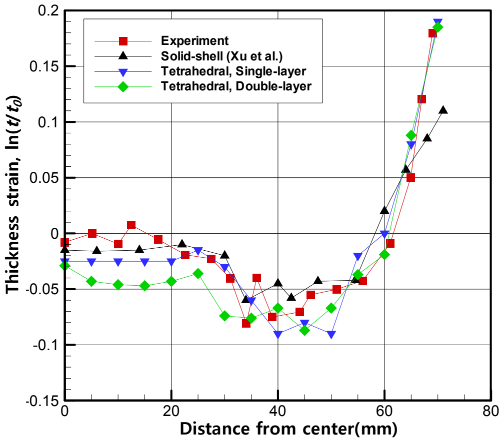
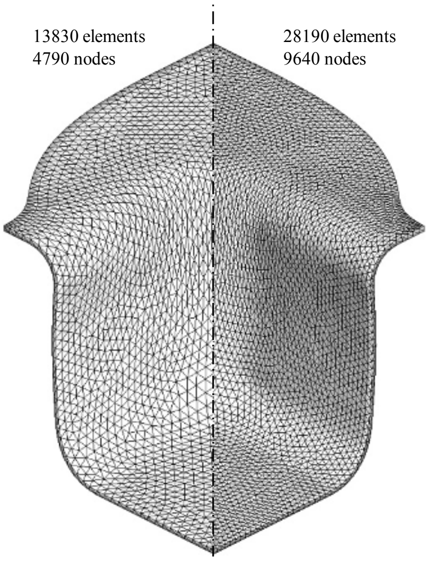


Fig. 11. Variation of thickness strain along the OA line.

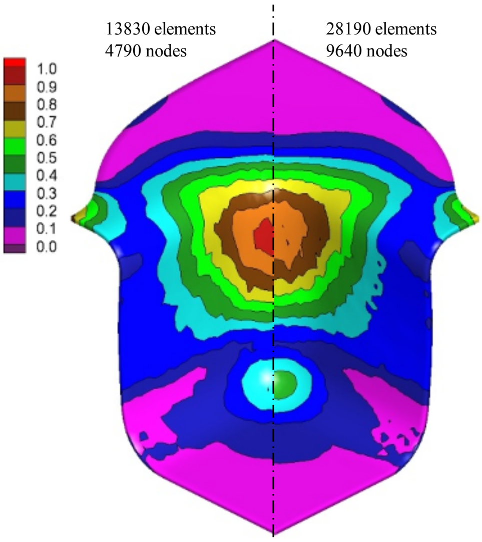
predictions of Xu et al. [5], and with predictions obtained using LS-DYNA3D by Mattiasson and Strange [12] (see Table 1). It can be concluded that the single- and double-layer tetrahedral finite element mesh systems yielded nearly the same predictions in terms of the overall deformed shape, and those predictions were in good agreement with both experimental results and other predictions.

Fig. 11 compares the predicted thickness strain (i.e., $\ln(t/t_0)$ where t and t_0 are current thickness and initial thickness, respectively) of the workpiece along the OA line with corresponding experimental results [14]. This comparison also confirms that the predictions were in good agreement with the experiments, indicating the feasibility of using the proposed technique to predict sheet or plate metal forming processes. The thickness strain shows slight bi-axial extension under the square punch. Around the punch corner, the minimum thickness strains were found. In the wall, as the distance from the center increases, the thickness strain increases. In the flange area, the thickness strain rapidly increases and finally shows the maximum value at the outer edge. Compared to the predictions of Xu et al., our predictions are better on the edge side, while the opposite is true on the center side. Of course, our underestimated predicted thickness on the center side resulted from the rigid plasticity assumption for the workpiece, since the center side was exposed to the elastic region for a very long stroke. Note that the tension stresses in the elastic region inevitably cause the thickness to be underestimated when simulating sheet metal forming processes via the rigid-plastic finite element method.

To reveal the mesh size effect, the same problem was simulated using a finer single-layer mesh system, that is, a single-layer mesh system composed of 28190 elements with 9640 nodes which are almost twice those of the coarser mesh in Fig. 9. The predictions were compared with those of the coarser mesh system seen in Fig. 8(a) in Table 2 and in Fig. 12. It can be seen that there exists little difference in deformation and effective strain between the fine and coarse meshes, which can



(a) Deformation



(b) Effective strain

Fig. 12. Comparison of deformation and effective strain between fine mesh and coarse mesh.

be negligible in the some engineering aspects.

4. Conclusions

In this paper, a plate metal forming process and a sheet metal forming process were simulated in a manner analogous to bulk metal forming simulations. A three-dimensional rigid-plastic finite element method with conventional linear tetrahedral MINI-elements was employed. The plate metal forming process was simulated using an intelligent remeshing capability optimized for bulk metal forming simulation, showing that the predictions are numerically acceptable but that special multi-layered mesh system would be better to represent the bending deformation, which is important in sheet metal forming.

Multi-layer tetrahedral mesh systems were thus considered for simulation of the sheet metal forming process. Single- and double-layer mesh systems were investigated, with particular attention to their effect on the deformed shape of the workpiece. The procedure was applied to the problem of the NUMISHEET93 international benchmark. The resulting predictions were compared with experimental results and other predictions found in the literature, and a qualitative agreement was noted. The number of layers had little influence on the overall deformed shape of the workpiece in the test example, but it might have a non-negligible influence on plastic deformation near corners, depending on the situation. The difference between the present predictions and the experiments was inevitable because of the assumption of fixed binder and the limitation of rigid-plasticity. It can be, however, noted that the effect of number of layers in the thickness direction is meaningful.

The comparison indicates the feasibility of applying conventional bulk metal forming simulators (with a little modification) to sheet or plate metal forming processes. These include hot sheet metal forming processes, as well as thick sheet or plate metal forming processes. It is believed that the solid element approach to plate or sheet metal forming will be one of mainstreams in the field of sheet metal forming in the near future and that this study is meaningful particularly in this context.

Acknowledgment

This research was financially supported partially by the Ministry of Education, Science Technology and National Research Foundation of Korea (NRF) through both the Human Resource Training Project for Regional Innovation and the Basic Science Research Program (NRF-2010-0025432).

References

[1] M. L. Wenner, Overview-simulation of sheet metal forming, NUMISHEET, *AIP Conf. Proc.*, Detroit, Michigan, USA (2005) 3-7.

[2] K. Roll, T. Lemke and K. Wiegand, Possibility and strategies for simulations and compensation for springback, NUMISHEET, *AIP Conf. Proc.,* Detroit, Michigan, USA (2005) 295-302.

[3] J. W. Cho, D. Y. Yang and W. J. Chung, A simplified approach for incorporating thickness stress in the analysis of sheet metal forming using shell elements, *Int. J. Numer. Methods Eng.,* 53 (10) (2002) 2311-2327.

[4] R. P. R. Cardoso and J. W. Yoon, One point quadrature shell element with through-thickness stretch, *Comput. Meth. Appl. Mech. Eng.,* 194 (9-11) (2005) 1161-1199.

[5] H. J. Xu, Y. Q. Liu, Z. B. Zhang and T. Du, Solid-shell finite element method for progressive die forming simulation,

Proceedings of the 13th international conference on metal forming, Toyohashi, Japan (2010).

[6] M. P. L. Parente, R. A. Fontes Valente, R. M. Natal Jorge, R. P. R. Cardoso and R. J. Alves de Sousa, Sheet metal forming simulation using EAS solid-shell finite elements, *Finite Elem. Anal. Des.,* 42 (13) (2006) 1137-1149.

[7] L. F. Menezes and C. Teodosiu, Three-dimensional numerical simulation of the deep-drawing process using solid finite elements, *J. Mater. Process. Technol.,* 97 (1-3) (2000) 100-106.

[8] M. C. Lee, W. J. Chung, J. G. Eom, J. S. Kim, K. S. Kim and M. S. Joun, Rigid-plastic finite element analysis of a cold plate forging process using tetrahedral elements, *Proc. Kor. Soc. Tech. Plast. Conf.,* Daegu, Republic of Korea (2010) 224-247.

[9] D. N. Arnold, F. Brezzi and M. Fortin, A stable finite element for the stokes equations, *Calcol,* 21 (4) (1984) 337-344.

[10] M. C. Lee, S. H. Chung, S. M. Jang and M. S. Joun, Three-dimensional simulation of forging using tetrahedral and hexahedral elements, *Finite Elem. Anal. Des.,* 45 (11) (2009) 745-754.

[11] M. S. Joun, M. C. Lee and J. G. Eom, Intelligent metal-forming simulation, *ASME, International Manufacturing Science and Engineering Conference(MSEC2011),* Corvallis, Oregon USA (2011) 50128.

[12] A. Makinouchi, E. Nakamachi, E. Onate and R. H. Wagoner, Numerical simulation of 3-D sheet metal forming processes, *Proc. of NUMISHEET' 93*, eds., Isehara, Japan (1993).

[13] M. C. Lee, M. S. Joun and J. K. Lee, Adaptive tetrahedral element generation and refinement to improve the quality of bulk metal forming simulation, *Finite Elem. Anal. Des.,* 43 (10) (2007) 788-802.

[14] J. Dankert, Experimental investigation of a square-cup deep-drawing process, *J. Mater. Process. Technol.,* 50 (1-4) (1995) 375-384.

ManSoo Joun is a professor of Gyeongsang National University in South Korea. He graduated from Seoul National University for his bachelor's degree, from KAIST for his master's degree and from POSTECH for his Ph.D. degree. He has developed and commercialized a general-purpose metal forming simulator called AFDEX, which is competitive in terms of solution accuracy and user-friendliness.