ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

FINITE ELEMENT SIMULATION TO PREDICT WRINKLING IN LOW CARBON STEEL DEEP DRAWING PROCESS USING ISOTROPIC MODEL

M. N. Iryana and M. Jasri
Faculty of Mechanical Engineering, Universiti Malaysia Pahang, Pekan, Pahang, Malaysia
E-Mail: iryana_yana@hotmail.com

ABSTRACT

Deep drawing is one of important sheet metal processes used in manufacturing industry. It is a process for shaping sheet metals into cup-shaped products. Prediction of forming results can help to save material and production time. Hence, the finite element (FE) simulation provides the best answer to predict the early defects of forming product. This paper describes the use of ABAQUS/ Standard FE simulation in a square cup deep drawing process using low carbon steel sheet metal. Here, the isotropic properties were used for the material's input data in the FE model and then, the result was validate via experimental product. The objective of this paper is to study the effectiveness of FE model to predict the wrinkling defect in the deep drawing process. As the result, the FE model predicted the similar wrinkling pattern with the experimental product.

Keywords: deep drawing, finite element simulation, isotropic properties.

INTRODUCTION

Deep drawing is one of the sheet metal forming processes which is widely used in manufacturing industry [1]. It is considered as deep drawing when the depth exceeds the initial thickness. In the process, the blank is deformed by an action of a punch forcing the metal into the die cavity. The products usually have a complicated shape, thus undergoing several successive operations are needed to obtain a good quality product. In order to produce defect free product, it is necessary to control every single parameter during the production. Some of the parameters that need to be control are initial blank geometry, surface between tool's interference and surface deflection. Hence, numerical method using FE simulation provides the best solution to control the parameters [2]. By using this method, the prediction of final product can be obtained. This can significantly reduce the production costs and also reduce the lead time to manufacture the products.

Generally, the FE simulation includes an input to simulate the model, a solution and an output which can be considered as a product prediction. To simulate a FE model, it is necessary to have accurate information on the geometry of the tooling, the sheet metal properties, the friction coefficient of the tools and the sheet interface. In this present paper, the reliability of isotropic hardening model is investigated. This is being done by using the isotropic properties as material input to simulate deep drawing process. The software used for the simulation is Abaqus with static/general analysis. The material used for the investigation is 1.5mm of thickness low carbon steel. The blank size is 100mm x 100mm. The die size is 50mm x 50mm with edge radius 2mm which is actually the desired square hole. The punch size is 44mm x 44mm with 2mm edge radius. The dimensions for FE tool's geometries are similar to the dimension experimental tool's geometries.

FINITE ELEMENT MODEL

The finite element model consists of tools and blank. The tools are punch and die and the blank is the sheet metal. The tools are designed as rigid whereas blank is deformable part. The element used for the tools is 4-node, three-dimensional rigid surface elements (R3D4) and for the blank is 4-node, bilinear finite-strain shell element (S4R). The coefficient of friction between punch and blank and between die and blank is 0.125. The value is taken from study conducted by Ayari and Bayraktar [3] as an optimum coefficient of friction to simulate deep drawing process. One quarter sector of the tools and blank have been constructed for the simulation model to reduce to computational time.

The material was modelled as an elastic-plastic material with isotropic elasticity which defined by Voce law theory as in Equation 1.

$$\sigma = \sigma_0 + Q(1-e^{-b\bar{\varepsilon}}) \quad (1)$$

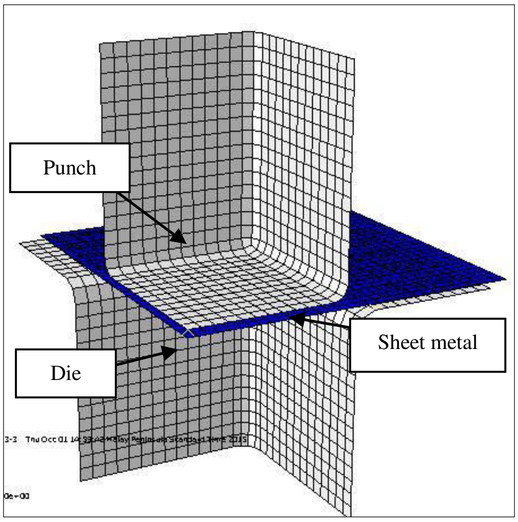
where σ_0 is a yield stress, Q is the maximum change in the size of the yield surface and b is the rate at which the maximum reached. The parameters for input material properties are acquired from tensile testing as shown in Table-1.

Table-1. Low carbon steel properties

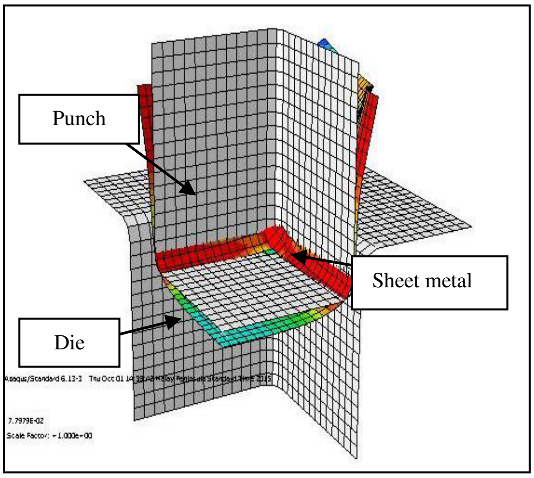
Young Modulus (Mpa)	**207000**
Poisson ratio	0.3
Yield Strength (MPa)	190.22
Q (Mpa)	726.324
b	120.031

Figure-1 shows the FE model at the initial stage, loading stage and unloading stage.

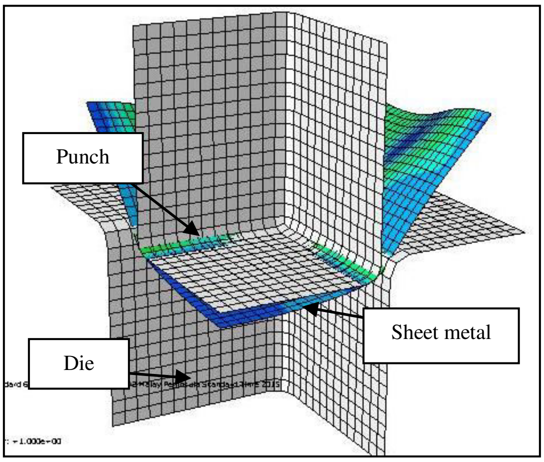
ARPN Journal of Engineering and Applied Sciences



(a)



(b)



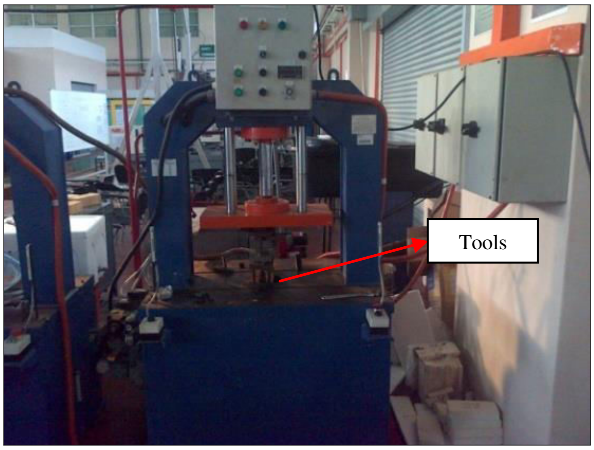
(c)

Figure-1. Finite element model; (a) Initial stage, (b) Loading stage, (c) Unloading stage.

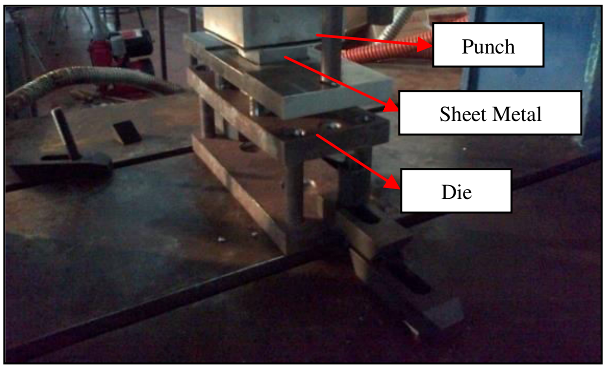
The boundary condition for the blank is set as asymmetrical at x and y axis whilst the die part is set as ancestry where it is constrained and fix at any axis directions. The punch is assigned to move vertically downwards to deform the blank into the die cavity. There is no holding force applied in this simulation.

RESULTS AND DISCUSSIONS

The FE result is compared to investigate its reliability and deep drawing process was carried out experimentally. Figure-2a and 2b show the tools and the hydraulic press machine that have been used.



(a)



(b)

Figure-2. Deep drawing experiment set up; (a) Hydraulic press machine, (b) Tools used in the experiment.

Figure-3 shows the experimental product from the process. The low carbon steel sheet metal was successfully drawn for about 8mm depth into the die cavity.

ARPN Journal of Engineering and Applied Sciences

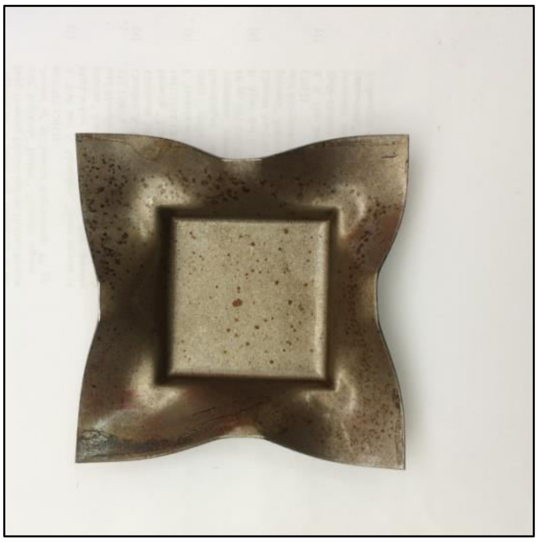
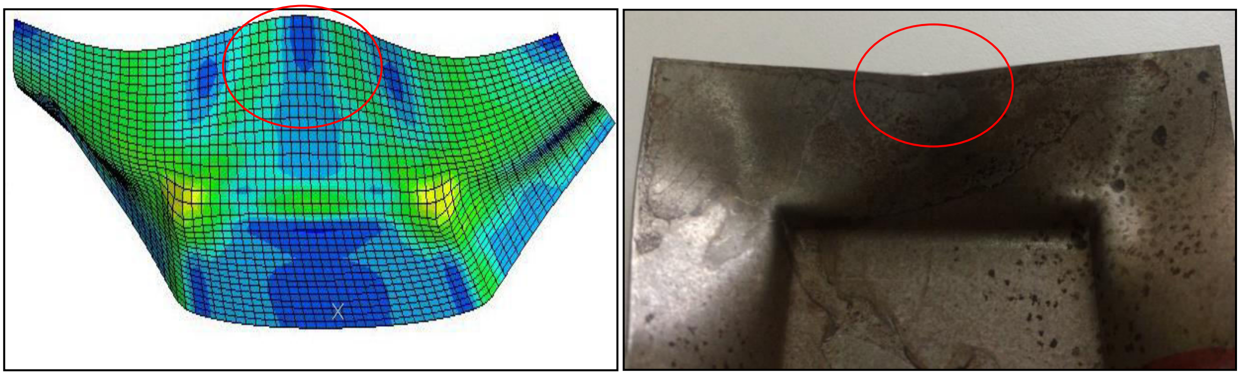
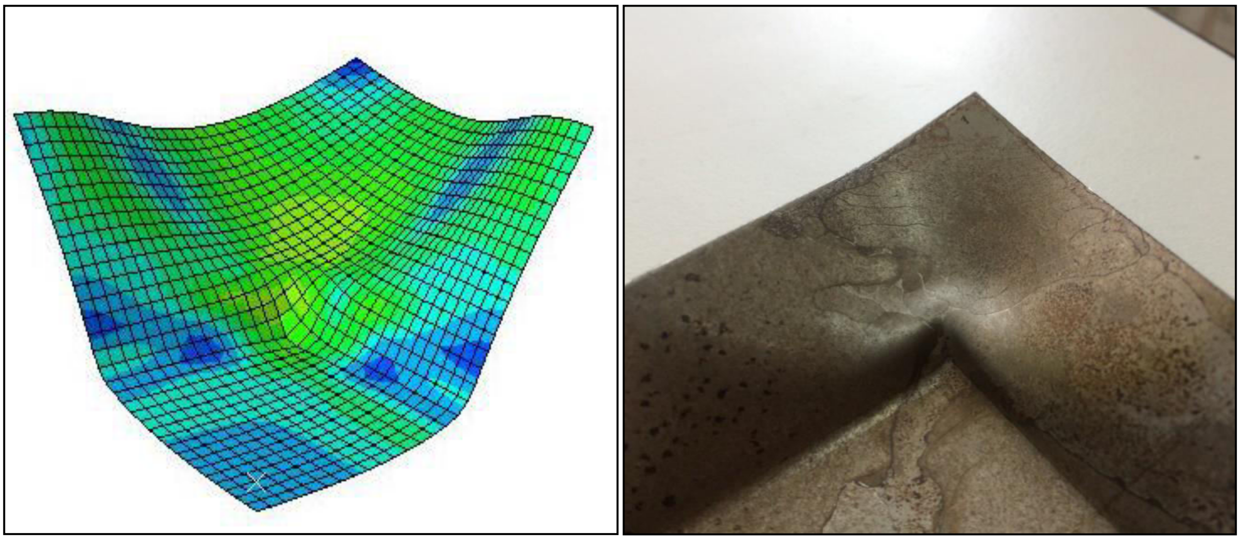


Figure-3. Experiment product.

Figure-4a and 4b show the comparison result for both simulation and experimental results. A qualitative good agreement between the experiment and the simulation has been shown throughout the figures. There is a concave up at the middle of both products as circled in the figures. The shape of concave up means the wrinkle formation which occurred along the flange of the cup. The blank rises towards the side wall of punch with the formation of wrinkling along the flange. This happens due to zero holding force applied into the sheet metal. When the holding force is less than needed, there is a big tendency for the wrinkles to occur [4]. The wrinkling also occurred due to tangential stress acted onto the sheet metal when it was drawn into the die cavity. The greater die cavity depth, the blanker will pulled down into the die cavity, the larger tangential stress occurred and the greater risk of wall wrinkling will occur [3].



(a)



(b)

Figure-4. Simulation and experimental results compared; (a) half deformable part, (b) one quarter deformable part.

CONCLUSIONS

As a conclusion from this study, the FE result was compared to the experimental result visually for any similarity. The isotropic model in FE simulation was successfully predicted the wrinkling defect with zero holding force applied. It was validated by the similar wrinkling pattern shown in the experiment process. To improve the FE simulation in representing the real deep

ARPN Journal of Engineering and Applied Sciences

drawing forming process, it is necessary to consider a constitutive model that takes into account the bending and unbending condition. For this purpose, kinematic and mixed hardening model have been acknowledged to provide the answer. Another study is already in final stage to test the reliability of kinematic and mixed hardening model in predicting the wrinkling behaviour in deep drawing process and the result will be published in another publication.

ACKNOWLEDGEMENT

The authors are indebted to RDU141004 and GRS 1403181 research grants for the financial support.

REFERENCES

[1] L. Chen, J. C. Yang, L. W. Zhang, and S. Y. Yuan, "Finite element simulation and model optimization of blank holder gap and shell element type in the stamping of a washing-trough," Journal of Materials Processing Technology, 182, pp. 637-643, (2007).

[2] A. G. Mamalis, D. E. Manolakos, and A. K. Baldoukas, "Simulation of sheet metal forming using explicit finite-element techniques: effect of material and forming characteristics Part 1. Deep-drawing of cylindrical cups," Journal of Materials Processing Technology, 72, pp. 48-60, (1997).

[3] F. Ayari and E. Bayraktar, "Parametric Finite Element Analysis for a square cup deep drawing process," Journal of Achievements in Materials and Manufacturing Engineering, 48, pp. 64-86, (2011).

[4] S. S. Han and H. Huh, "Modified-membrane finite-element simulation of square cup drawing processes considering influence of geometric parameters," Journal of Materials Processing Technology, 48, pp. 81-87, (1995).