Finite Element Method for Sheet Metal Forming Analysis of Cover Fuel Filler

Phornchai Kongwattanachai[1] Chulsiri Sringamphong[2]
Dilok Sriprapai[3] Dech Budcharoentong[4]
Department of Tools and Materials Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Bangkok, 10140, Thailand
Tel: 0-24709312 Fax: 0-28729080 Email: itgv1@ksc.th.com[1], chulsiri.sri@kmutt.ac.th[2], dirok.sri@kmutt.ac.th[3], dech.bud@kmutt.ac.th[4]

Abstract

This study is the sheet metal forming experiment and analysis which uses finite element method to analyze Cover Fuel Filler automotive part with at least wrinkles in draw-die process. The study has emphasized by using variant type of draw bead on die face to control the metal flow. The proper draw bead condition for die set and experimental Cover Fuel Filler forming has been corrected. After that the wrinkles and the thickness along two cross sections of the experiment and analysis one have been compared. The final wrinkle geometry from finite element method is consistent with the experiment and the maximum error of each thickness profile is 1.48 and 6.75 percent only. Therefore we can conclude that the finite element method with proper draw bead condition can be used to minimize a wrinkle of Cover Fuel Filler forming effectively.

Introduction

Cost and time of sheet metal forming process in automotive parts industry can be reduced by using finite element method or FEM. Generally, a great deal of time and cost of automotive tooling is consumed by finding the appropriated tool geometry and manufacturing or try-out tooling process. Try-out process can be represented by FEM which can simulate and predict the forming result such as deformation, fracture, wrinkle, thickness or stresses. The defect of forming can be found and corrected with FEM before actual forming by redesign conditions of tooling such as die-face geometry. The study of this paper is to simulate and find out the proper die-face condition to success forming of Cover Fuel Filler part which is an inner automotive part. The results presents here are from the analysis and compare of final deformation and thinning.

Objectives

The objective of the study is to apply FEM in sheet metal forming. The primary focus is to correct sheet metal tooling and to compare the FEM output with actual forming part.

Approach

The Cover Fuel Part is an automotive part which is used FEM to analysis. The part is the first draw stage in conventional stamping process of which involved a complicated three-dimensional deformation. The geometry of this part is show in Figure 1.

Figure 1. Cover Fuel Filler part

This paper will be organized in 5 steps. The first step is to specific the material properties of sheet metal. The second step is to define tools positioning. Next step is to specific the yield criterion with the involved finite element constitutive equations. After that FEM result is use to investigate the cause of problem. The final step is comparing the FEM final output after to correct die-face geometry with the actual stamping part.

1. Material properties of sheet metal

Material properties of sheet metal for FEM are extremely importance. There are two characteristics of material that are stress-strain relation in plastic region and anisotropic property due to rolling in manufacturing process of sheet metal. Both properties can be identified by tensile testing method. Stress-strain relation is represented by power law function which is expressed by the following:

$$\sigma = K\varepsilon^n \quad (1)$$

Where K is the Strength Coefficient (N/mm[2],MN/m[2])

n is the Strain-Hardening Exponent.

Another is anisotropic property. In some cases, Anisotropic property rather effect to the formability of sheet metal forming. During forming, it usually exhibit two different forms. One is concerned with the hardening behavior when measured along different directions on the plane of the sheet. It means that the relationship of the stress and strain is different in different directions. Another anisotropic property is the different thinning values when measured along the plane of the sheet instead of through the thickness direction. Anisotropic property of sheet metal can be measured by plastic strain ratio or R-value which is expressed by the following:

$$R = \frac{\ln(w_0 / w_x)}{\ln(t_0 / t_x)} \qquad (2)$$

Where, w_0 , w_x are width of previous and after of tensile testing specimen. t_0 , t_x are thickness of previous and after of tensile testing specimen respectively.

A sheet material used in the experiment is SPCC with 0.8 mm. thickness. The material properties are listed in Table 1.

Table1. The properties of SPCC 0.8 mm.

Parameter	Notation	Value
Strain-Hardening Exponent	n	0.473
Strength Coefficient	K	444.38
Plastic Strain Ratio	R	2.43

2. Tools positioning

Tools positioning included Die, Punch and Blank Holder are illustrated in figure 2. In conventional stamping process, Die move downward to Blank Holder and Punch to deform Blank sheet respectively.

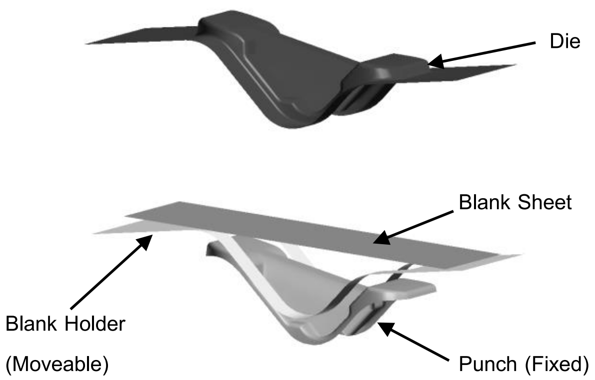

Figure 2. Tools positioning

3. Yield criterion and finite element constitutive equations

The yield criterion for this experiment uses Hills' (1948). This criterion is suitable for plane stresses problem with anisotropic behavior of steel metal sheet which is expressed by the following:

$$F(\sigma) = \sigma_i = \left[\sigma_{11}^2 + \sigma_{22}^2 - \frac{2R}{R+1} \sigma_{11}\sigma_{22} \right]^{1/2} \qquad (3)$$

Where σ_i is effective stress

σ_{11} and σ_{22} are the principle stresses

Figure 3 illustrate Hills' (1984) yield locus with different R-value under complex stress components with the same effective stress (σ_i). In isotropic material, without plastic strain ratio effected or R=1, the effective stress in which can deform material equal σ_i in both principle stress axis of plane stress problem ($\sigma_{11} = \sigma_{12} = \sigma_i$) . If R-value of material is higher than 1 or is anisotropic material, In case of tensional plane stress problem, it mean that the effective stress must clearly more than σ_i of both principle stress axis $(\sigma_{11} = \sigma_{12} > \sigma_i)$.

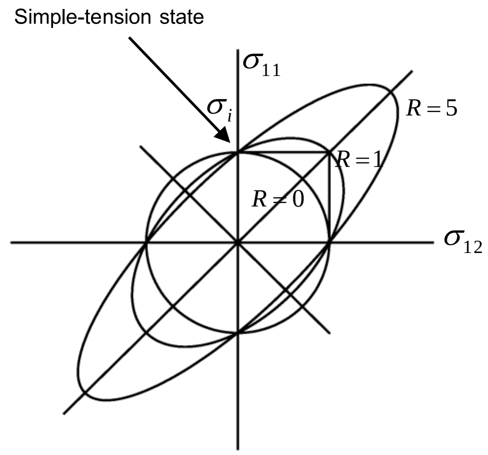

Figure 3. Stress space with different R-values

The constitutive finite element formulations in metal forming which involve the stress-strain relationships can be determined in the general form of force and stress by following equations:

$$F_i = \sigma_{ij} n_j \qquad (4)$$

Where F_i is external force

σ_{ij} is stress of element

n_j is direction which caused by force

We can device the stress in eq. (4) into mean stress or hydrostatic stress (σ_m) and deviatoric stress (σ'_{ij}). The deviatoric stress is the only stress that lead element to deform.

$$\sigma_{ij} = \sigma'_{ij} + \delta_{ij}\sigma_m \qquad (5)$$

The basis constitutive finite element formulation in metal forming can be derived in form of:

$$\delta\pi = \int_v \bar{\sigma}\delta\dot{\bar{\varepsilon}}dV + \int_v \lambda\dot{\varepsilon}_v dV - \int_s F_i\delta w_j dS = 0 \qquad (6)$$

Where δ is Kronecker Delta matrix

λ is Lagrangian Multiplier

π is total potential energy

$\dot{\varepsilon}$ is strain rate

w_j is the velocity on element

V and S is volume and surface area of material

4. FEM result

Due to tools have been already made and met the wrinkle problem which show in figure 4. FEM program ,Dynaform-PC, which is a commercial FEM program has been used to investigate the wrinkle problem. The result of computing found that Die has touched the Blank sheet before Blank Holder as show in figure 5. This situation makes Blank sheet bending while being deformation. The corrected process is that Die must press Blank holder first and move down to Punch to form Blank sheet respectively.

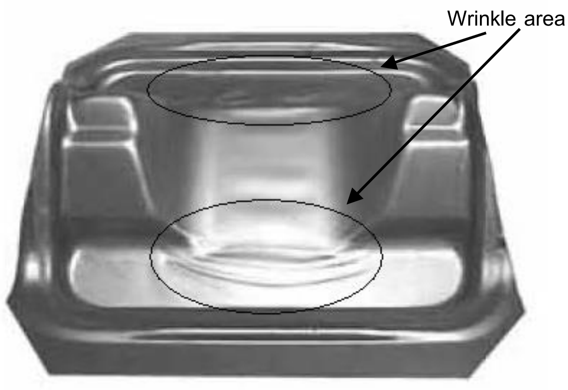

Figure 4. The wrinkle problem of part

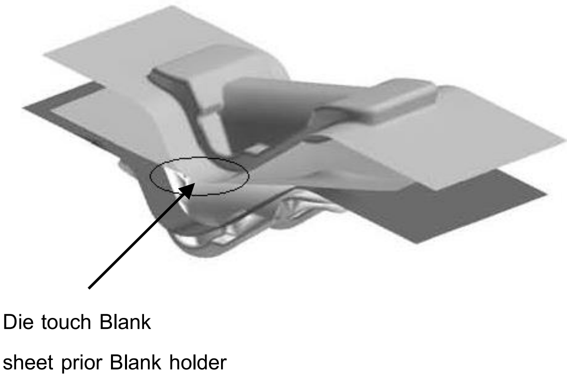

Figure 5 Simulation of wrinkle cause

Figure 6 show the result of the actual Blank sheet while deforming process. Figure 7 is the simulation form FEM that show bending and starting state of wrinkle problem.

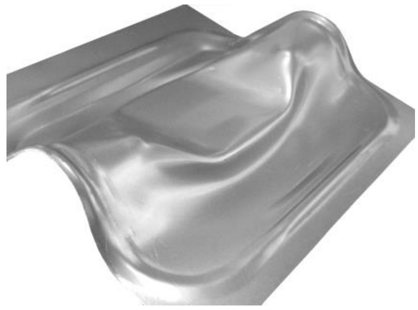

Figure 6 Actual wrinkle part while deformation

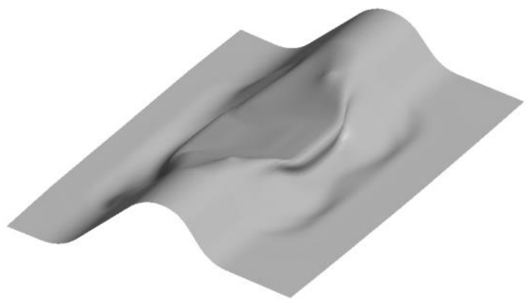

Figure 7 FEM simulation of starting wrinkle

To correct the wrinkle problem, Die face has been redesigned by creating a draw bead to control flow of metal where located on two side of tool as show in figure 8. The purpose of this draw bead is to stretch Blank sheet back for reducing the wrinkle cause. After trying with various types, The final Draw-bead is to correct on tools. The FEM simulation with redesigned Die face is show in figure 9 which consist of experimental part as show in figure 10.

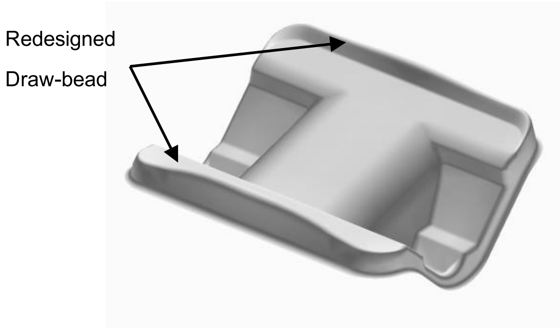

Figure 8 Redesigned Draw bead

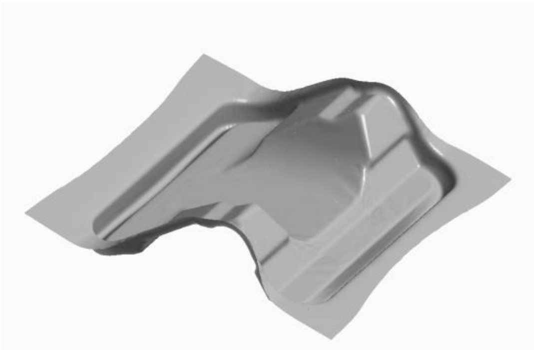

Figure 9 FEM simulation with redesigned draw bead

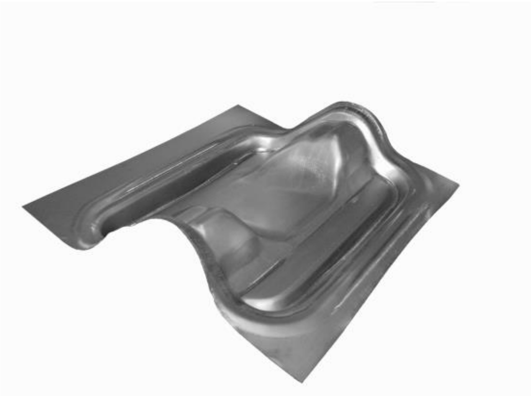

Figure 10 Experimental part with redesigned draw bead

5. Comparison FEM output with experimental part

The comparison of FEM output and experimental part after corrected tools has investigated the thickness along cross-section where are critical locations. This location has been deeply deformed in vertical direction particularly the first cross-section. The second cross-section has been also compared due to effect of correcting wrinkle problem with redesigned die face geometry. The thickness profile and the percentage of deviation comparison of both cross-sections are located on profile no. 1 and no. 2 as show in figure 11.

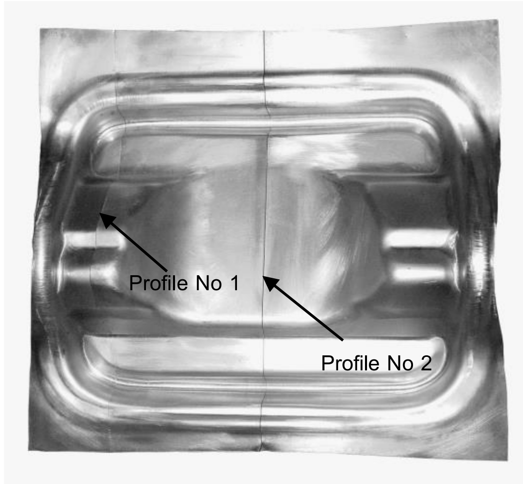

Figure 11 Cross-section comparison profiles

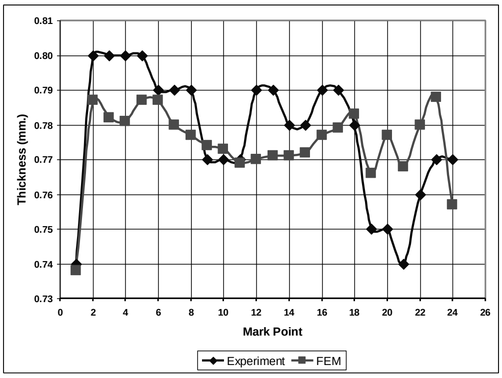

Figure 12 Thickness profile comparison of cross-section no.1

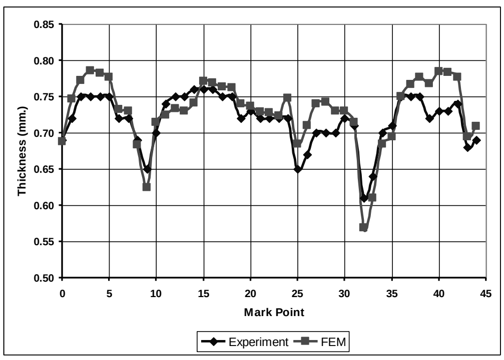

Figure 13 Thickness profile comparison of cross-section no.2

Figure 12 and 13 are thickness comparisons of FEM and experimental part along cross-section no.1 and 2 respectively.

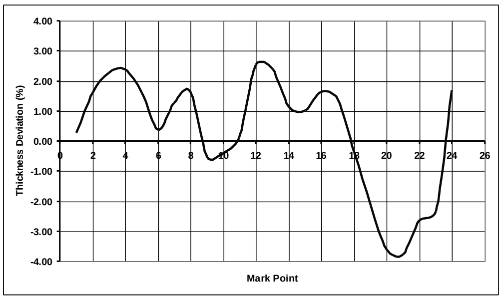

Figure 14 Percentage of thickness deviation of cross-section no.1

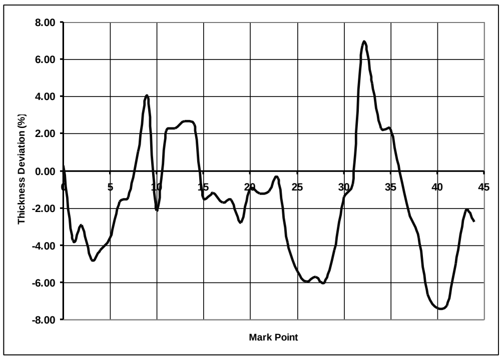

Figure 15 Percentage of thickness deviation of cross-section no.2

Figure 15 and 16 illustrate percentage of thickness deviation. The result indicates that maximum deviation of cross-section no.1 is only 3.85 percent and no.2 is also 5.29 percent respectively.

Discussion and Conclusion

The analysis of Cove Fuel Filler stamping has objective to apply finite element method in stamping problem operation. The FEM accuracy is depended on the completetion and accuracy of input data which used in FEM computation such as material and machine information. In this study, the FEM result is consist of experiment that illustrates in thickness deviation comparison. However FEM is not only for simulating the deformation of sheet metal but it can also investigate the cause of problem which difficultly investigates in actual operation. Therefore FEM can help tools designer and engineer who design and correct the tools effectively. The production of tooling has been reduced in time operation. Cost production may has also been reduced by using FEM for tools try-out.

The experiment of analysis one can significantly correct the wrinkle problem. The cause of wrinkle problem has been found by the result of simulation that is Die touch Blank Sheet prior Blank Holder. Even though tooling set has been produced, FEM can also investigate and correct the problem with redesigning the proper draw-bead geometry. However, FEM can effectively apply to sheet metal stamping process before actual tools production. Moreover FEM is also suitable to analysis of multi-stage stamping or compound stamping tools.

References

[1] Fromentin, S., Martiny, M., Ferron, G., Tourki, Z., Moreira, L.P. and Ferran, G., 2001, “Finite Element Simulations of Sheet-Metal Forming Processes for Plannar-Anisotropic Materials”, International Journal of Mechanical Sciences, Vol. 43, pp. 1833-1852.

[2] Rowe, G.W., 1991, Finite-Element Plasticity and Metal Forming Analysis, Cambridge University Press, New York, pp. 25-134.

[3] Kobayashi, S., Oh, S. and Altan, T., 1989, Metal Forming and the Finite Element Method, Oxford University Press, New York, pp. 58-109.

[4] Dieter, G.E. and Bacon, D., 1988, Mechanical Metallurgy, McGraw-Hill, London, pp. 17-102, 275-288.