

Dome Forming Study on Polypropylene Aluminium Laminate

Wentian Wang

Research School of Engineering, Australian National University
Canberra, Australia, +61 433 170 121
wentian.wang@anu.edu.au

Shankar Kalyanasundaram

Research School of Engineering, Australian National University
Canberra, Australia, +61 2 6125 8369
shankar.kalyanasundaram@anu.edu.au

Abstract—This work investigates the forming behavior of polymer metal laminate (PML) in stamp forming through both experiment and finite element modeling. The effect of blank holder force on the PML forming was determined by comparing the strain evolution under different blank holder forces. A finite element model which is capable of simulating forming process of PML was developed and then validated by the results obtained from experiments. This model predicts PML forming behavior accurately and will provide a fundamental computational framework for widespread use of this advanced material system in automotive applications

Keywords—component; Polymer Metal Laminate; Stamp Forming; Finite Element Modeling; Blank Holder Force

I. INTRODUCTION

In 2008 personal vehicles consumed as much as 21,395TWh energy globally, which accounted for up to 16% of total carbon emissions [1-2]. This study outlines the solution to one of the more increasing problems of reducing the energy consumption of private vehicles by replacing steel with PML. PML is in a sandwich structure with two pieces of metal outer layers and one piece of polymer at center core. PML has great potential to replace steel due to its lighter weight yet similar stiffness properties [3]. This paper investigates the behavior of PML in stamp forming which is a common method used in the manufacturing industry to mold material into any desired shape. Our previous studies [4-7] have shown that blank holder force, feed-rate and temperature have important influence in stamp forming of laminate material systems. A novel real time strain measurement system (ARAMIS) was used to capture strain evolution during forming. This is world first for this class of material system and an open die was used to acquire the strain evolution during forming. A finite element model which is able to accurately simulate the forming behavior of PML in stamp forming was developed in this study and the finite element model was then validated by the experimental results.

II. EXPERIMENTAL PROCEDURE

A. Material Preparation

The polymer metal laminate structure analyzed in the project consisted of two 0.6mm 5005 H34 Aluminum outer layer sheets and 0.8mm plain polypropylene sheet in between. The aluminum was etched before gluing with polypropylene to

eliminate the material impurities as well as surface dirt. Two hot-melt polypropylene adhesive layers were employed to bond the polymer metal laminate used in experiments. In the gluing process, two pieces of aluminum sheets, two adhesive layers and one piece of polypropylene were stacked together in a sandwich structure before been placed in the heat press machine. Samples were then heated up to 155°C which reaches the bonding temperature of the adhesive layers but is less than the melting point of polypropylene. A pressure of 300kPa was then applied on the samples for 2 minutes. Afterwards, samples were cooled down to 80°C at a rate of 50-70°C/min before removal. The pressure was maintained during cooling to make sure that PML was bonded properly. All test samples were cut into a circular shape with a diameter of 180mm by water-jet cutting which was controlled by computer to achieve precision in cutting.

B. Experimental Setup

The stamping press was controlled through a hydraulic feed controller, the dynamics of which was provided by a 20-liter accumulator, charged using a two stage pump located next to the machine. All test conditions including blank holder force, feed rate and post-forming holding time could be set up through the software installed in the computer integrated to the press machine. The open die design of the press enables ARAMIS three dimensional strain measuring system to capture the forming process during experiments. The ARAMIS system provides information on the surface deformation during forming and hence is useful in validating the computational model developed in this study. This system is based on three dimensional photogrammetry methodologies where two CCD cameras are used to capture the surface deformation. A stochastic pattern is applied to the sample surface and using a triangulation scheme, three dimensional deformations during forming could be obtained. This is a very accurate system for strain measurements where the error in the strain measurements is less than 0.02%.

III. FINITE ELEMENT MODELLING

The finite element analysis (FEA) work for this study was implemented by ABAQUS. The ABAQUS model created to simulate the forming behavior of PML in stamp forming uses implicit rather than explicit formulation, which means the equilibrium conditions were checked by the forming simulation

for each time increment. The supercomputing facility at the Australian National University was used to run all the simulations.

A. Model Geometry

Punch, die, blank holder and blank were designed in ABAQUS to match the experimental set up. The punch was modeled as a rigid body of cylinder with radius 50mm, 50mm for the hemisphere radius and 50mm for the top height. The die was modeled as a rigid body with a flat geometry of 102.5mm radius circle concentrically connecting with a cylinder of radius of 52.5mm. A fillet with radius of 12.5mm was designed in the connection between the die and cylinder. The blank was modeled using shell elements and was formed by stacking three layers of material (two aluminum outer layers and polypropylene layer in between) to a total of 2.2mm in thickness. The radius of each layer was 90mm and the thickness of aluminum and polypropylene layer were 0.6mm and 1mm respectively. Fig. 1 illustrates the finite element model used in this study.

B. Material Properties

There were two different constituents in the laminate structure analyzed in the project: H34 5005 aluminum and plain polypropylene. The selection of material property was crucial for accurately predicting the material behavior in stamp forming. Aluminum was modeled as an isotropic elastic-plastic material. The total strain decomposition can be expressed as:

$$\epsilon = \epsilon^{el} + \epsilon^{pl} \quad (1)$$

Where ϵ^{el} and ϵ^{pl} are elastic strain and plastic strain, respectively.

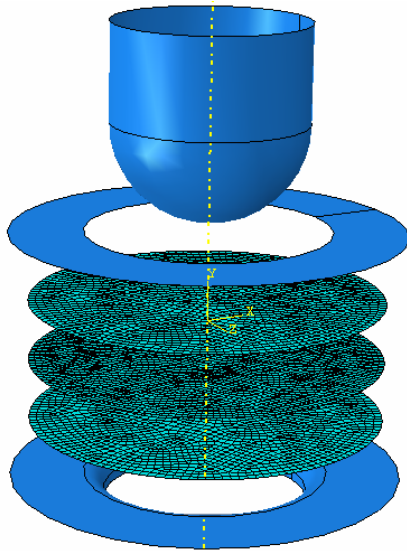


Fig.1: Finite element Model of Forming

The Young's Modulus and Poisson's Ratio used in the calculation were 69 GPa and 0.33, respectively.

The stress strain behavior of the aluminum in plastic region was obtained from the tensile tests and was given in a tabular form in ABAQUS model [8].

TABLE I. PLASTIC STRESS/STRAIN BEHAVIOR OF ALUMINUM

Stress (MPa)	41.4	86.8	105.4	118.4	128.6	137.2
Plastic Strain	0	0.03	0.06	0.09	0.12	0.15

The polypropylene layer was defined as linear elastic in this work with the material properties listed in table II.

TABLE II. POLYPROPYLENE MATERIAL PROPERTIES IN ABAQUS

Density (kg/m ³)	Young's Modulus (GPa)	Poisson's Ratio (%)	Layer Thickness (mm)
900	1.5	0.45	1

C. Load and Boundary Conditions

The only test condition variable set up in the experiments is named blank holder force (BHF) which is modeled as a constant force pressing the blank holder in FEA model. During the forming simulation, the die is fully fixed in all directions so that it stays stationary all the time. The blank holder is fixed in all directions except the vertical plane to allow the action of the blank holder force. The punch was set to move down constantly to 12mm in 2 seconds. There is no extra boundary condition to the blank since its position is constrained by the die and blank holder.

D. Contact Conditions and Mesh Specifications

A penalty contact condition was used to model the contact conditions with a coefficient of friction 0.3 defined for all the contact conditions in the finite element model. A finite amount of sliding is allowed in the slide formulation to facilitate material drawing. Tool geometries were defined as master surface as it is much more likely the tool will penetrate the blank than the other way around. Aluminum is expected to penetrate the polypropylene layer during the forming simulations since aluminum is much stiffer than polypropylene. Therefore, the aluminum layer and the polypropylene layer were defined as master surface and slave surface, respectively in the contact condition between them. For a similar reason, the polypropylene layer was assigned more elements with less element size (3,500, 3mm) than that of the aluminum layer (2,000, 4mm).

IV. RESULTS AND DISCUSSION

The failure depth of the PML system was found to be approximately 15mm and a depth of 12mm was chosen for comparison between experiments and FEA simulation.

A. Wrinkling and Drawing

In stamp forming, blank holder force plays a substantial role in minimizing wrinkling as well as fracture of samples. In forming experiments, PML was formed under 2kN, 7kN and 14kN BHF, respectively. At low BHF, there were significant wrinkling effects appearing on the out flange as failure mode. By increasing the BHF, the wrinkling phenomenon became

almost non-existent as shown in Fig.2. It matched with the findings reported by Gresham et al [7].

B. Experimental Results

An investigation on strain evolution was conducted at three different points: center point (point A), the unsupported edge 15mm away from the center (point B) and the unsupported edge 30mm away from the center (point C). For each individual point, the strain evolution was plotted and then compared to determine the influence of BHF on the forming behavior of PML in stamp forming. Fig.3 shows the comparison between each BHF trial at point A, B and C, respectively.

Strain at point A rises with a decreasing slope and flattens out at the peak value. Fig.3 (a) shows clearly that BHF affects the strain evolution of the pole region which experiences a sharper increase at low forming depth and a higher peak value at high forming depth under large BHFs.

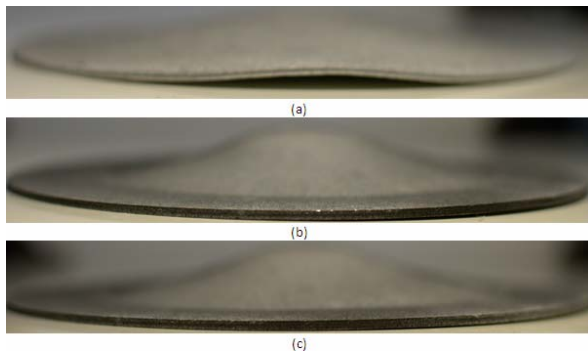
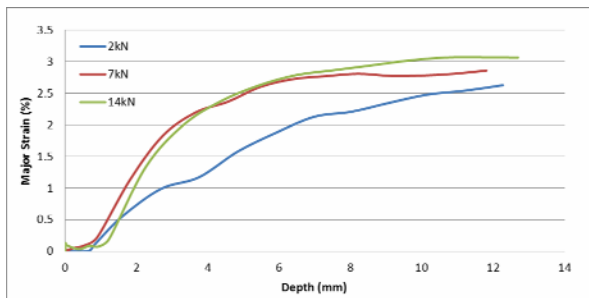
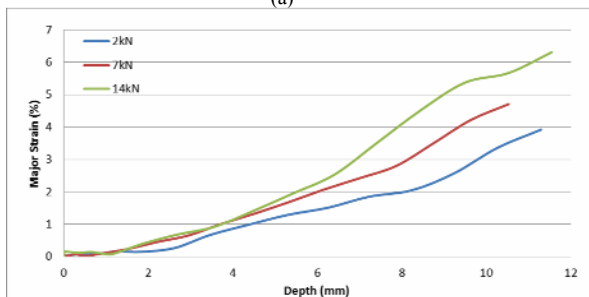


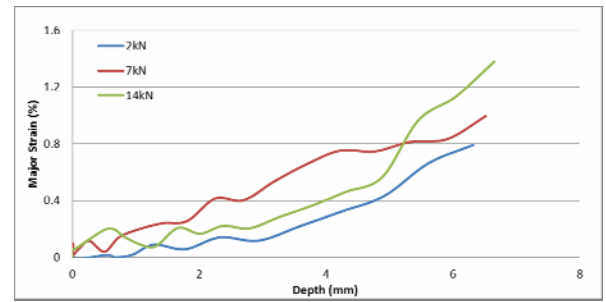
Fig.2: Front view of PML after the experiments at a. BHF 2kN b. 7kN, c. 14kN



(a)



(b)



(c)

Fig.3: Strain Evolution at point (a) A, (b) B, and (c) C, respectively.

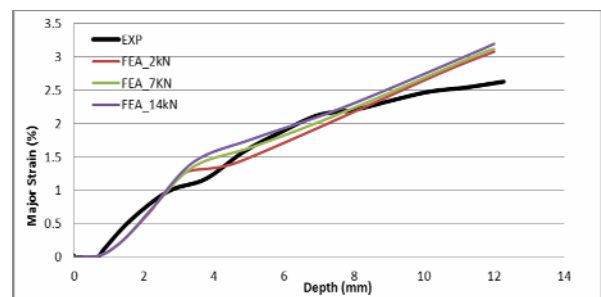
The strain evolution of point B, 15mm away from the pole, has been plotted in Fig. 3 (b) for each BHF trial. The major strain of this point rises with an increasing slopes to 3.9%, 4.7% and 6.3% for 2kN, 7kN and 14kN BHF, respectively. BHF has demonstrated a distinctive influence on the strain evolution of this point. It is worth noting that point B experiences a larger strain value than that of the pole after a certain depth, which indicates that the maximum strain shifts from the center to its nearby region as the forming depth increases. This phenomenon is caused by the friction between the punch and specimen, which is a common behavior for metallic alloys [9].

Fig.3 (c) shows the strain evolution at point C which is 30mm away from the pole. This point experiences a small strain throughout the whole forming process due to the contact conditions between the punch and the specimen. Due to the similar reason and also a small forming depth, BHF has a marginal effect on the strain evolution of this point.

It was found that BHF exerts certain influence on the strain evolution of the specimen center area, especially point B. A large BHF indicates more restrictions to the material from drawing in and hence large BHFs increase the stretch behavior of the PML in the forming. Thus samples experience a larger strain at high BHFs.

C. Validation of the Finite Element Model

An essential part of the study is to validate the FEA model by the results obtained from experiments. Experimental results at 2kN were used to compare with FEA simulations. Fig.4 shows the comparison between FEA simulation and experimental results at point A, B and C, respectively.



(a)

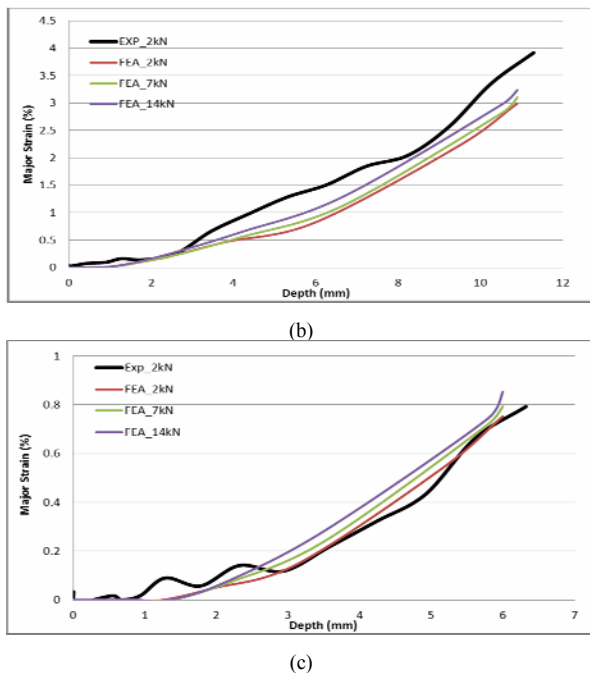


Fig. 4 Comparison between experimental results and FEA simulations at point (a) A, (b) B and (c) C, respectively

Fig.4 shows good agreement between experimental results and FEA simulations especially at point C which is the point experiencing the minimum amount of strain during forming. At the pole, FEA simulation fits well with experimental results prior to a depth approximate to 8mm and it overestimates the strain value afterwards. One possible explanation for this is that it is assumed that the polypropylene exhibits purely linear elastic behavior in finite element model and some discrepancy is expected when the PML experiences a high level of strain due to the nonlinear stress strain behavior. Based on the comparison at point B, there is an approximately constant difference between experimental results and FEA simulations. Since the maximum strain shifts from the pole to point B due to the friction between the contacts, one possible reason is that the friction condition between the punch and the sample does not predict the actual contact conditions precisely.

To further improve the accuracy of the finite element model, a more accurate friction condition is desired to eliminate the discrepancy. Also, it is necessary to obtain a more accurate constitutive behavior for polypropylene especially at large strains. This can then be implemented through a user defined material property routine in the finite element model.

CONCLUSION

This study has analyzed the behavior of PML in stamp forming through both experiments and a finite element model. The ARAMIS 3 dimensional optical strain measuring system was adopted to capture and compute the forming process of the PML in experiments. It was observed that the maximum strain shifts from the pole to the region close to it as the forming depth increases. Large BHF increases the stretch behavior of the material and PML experiences large strains at high BHF. The finite element model developed in this work has been proved to be able to simulate the forming behavior of PML in stamp forming accurately. The model predicts the strain evolution trends accurately with a certain discrepancy in values. To minimize the discrepancy, a more accurate friction condition between contacts and a user defined material property routine for polypropylene is desired.

The development of this finite element model simulating the forming behavior of PML is an important step to utilize stamp forming to produce such material. The findings of this research have developed an understanding of the stamp forming behavior of PML material system and in future will facilitate the application of this material system in the automobile industry.

REFERENCES

- [1] International Energy Agency, 2010, *2010 Key World Energy Statistics*, 9 rue de la Fédération 75739 Paris Cedex 15, France
- [2] T. Onoda and T. Gueret (2007), International Energy Agency. *Fuel Efficiency Road Vehicle Non-engine Components*. 9 rue de la Fédération 75739 Paris Cedex 15, France
- [3] Lutz et al. 1986, *High heat metal polymer laminate*, US Patent 4,601,941.
- [4] L. Mosse W. Cantwell M.J. Cardew-Hall P. Compston and S. Kalyanasundaram 2005 "Effect of process temperature on the formability of fibre-metal laminates", *Composites Part A: Applied Science and Manufacturing*, vol. 36, no. 8, pp. 1158-66.
- [5] L. Mosse W. Cantwell M.J. Cardew-Hall P. Compston and S. Kalyanasundaram 2006 "The influence of process variables on the formability of fibre-metal laminates", *Journal of Material Processing Technology*, vol. 172, pp. 163-168.
- [6] A. Sexton W. Cantwell and S. Kalyanasundaram 2012 "Stretch forming studies on fibre metal laminate based on a self-reinforcing polypropylene composite", *Composite Structures*, vol. 94, pp. 431-437.
- [7] J. Gresham, W.J. Cantwell, M.J. Cardew-Hall, P. Compston and S. Kalyanasundaram, "Drawing behavior of metal-composite sandwich structures", *Composite Structures*, 75(1-4):305-312, 2006.
- [8] Mosse, L., 2006, 'Stamp Forming of Fibre-Metal Laminates' PHD thesis, The Australian National University, Canberra, ACT, Australia.
- [9] C. Morrow, S. Dharmalingam, S. Venkatesan, S. Kalyanasundaram, *Stretch forming studies on thermoplastic composite*, In: Proceedings of 6th Australasian congress on applied mechanics, Perth, Australia December, 2010.