INFLUENCES OF DRAW FORMING PROCESS ON THE CRASH ANALYSIS OF A CIRCULAR CUP

R.M. Amman[1], M.F. Halim[2], D. Sivakumar[1],
I. Abu-Shah[2], S.N. Sulaiman[2], H. Samekto[1] and S. Subramonian[3]

[1]Faculty of Mechanical Engineering,
Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian
Tunggal, Melaka, Malaysia.

[2]Faculty of Technology Engineering,
Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian
Tunggal, Melaka.

[3]Faculty of Manufacturing Engineering,
Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian
Tunggal, Melaka.

Corresponding Author's Email: [1]sivakumard@utem.edu.my

Article History: Received 6 August 2017; Revised 19 October 2017;
Accepted 15 December 2017

ABSTRACT: The change of a structural part that occurred after forming process can affect crash response. Current industrial practice only utilizes the geometry in crash analysis. This study investigates the effect of forming histories of a circular cup formed by draw forming process in the crash simulation. Crash analysis at an initial velocity of 50km/h was performed using the explicit finite element code Radioss. The Johnson-Cook constitutive material model was used to characterize the material properties of advanced high strength steel DP600. Crash simulations are conducted in two different cases using a geometrical cup model with case 1 no forming history and case 2 all forming histories obtained from forming process. Results from this study indicate that the mechanical response of steel DP600 in a crash differ by 80.7 % for contact force and 5.87% for energy absorption when forming effects were considered. The contact force tends to increase more with displacement in case 2 compared to case 1. The non-uniform thickness and work hardening from forming process do alter significantly the crashworthiness of a structural part in the subsequent crash event.

KEYWORDS: *Draw forming; Circular cup; Crash analysis; Steel DP600*

1.0 INTRODUCTION

Sheet metal forming is a process which mostly are used to produce automobile parts that are difficult to join by welding. The development of Finite Element Model (FEM) for forming process has started to grow since the 1970s and gives exceptional achievement in metal forming research [1]. In order to produce defect-free stamped parts at the early design stage, engineers perform virtual experiments using numerical computation to predict the defects and improve the process without any physical prototyping parts and stamping tools [2]. Recently, this manufacturing process has become one of the most important processes in automotive industries. During the manufacturing process to form a designed shape, a thin blank sheet is subjected to plastic deformations using forming tools. The change of a structural part that occurred after forming process can affect subsequent process. The forming effect will become more significant on crash analysis since the structure experiences dynamic response. Common industry practice in crash simulation utilizes only the geometry of the structure without taking into consideration the changes in the physical properties during the forming process due to cost and time constraint [3].

Researchers had studied the effect of forming process to crash analysis. Dutton et al. [4] examine the crashworthiness of the side rail considering several forming results such as thickness variation, plastic strain and residual stress which is obtained from hydroforming analysis. Their study implied the important role of forming results to the crashworthiness prediction. Oliveira et al. [5] evaluated the crashworthiness of the aluminium alloy s- rail structure from the tube-bending process. They found that the change in geometrical features of s-rail with different bend radius play a significant role in offsetting any potential increases in force and energy absorption due to work hardening and thickness change in the material from the pre-bending. Moreover, Najafi and Rais-Rohani [6] also showed the importance of manufacturing effects on the crash response by using the multi-objective genetic algorithm approach. They conducted on sequentially coupled process-performance simulation for deep drawing analysis of magnesium alloy and concluded that the geometric attributes can significantly affect the energy absorption behaviour in which both maximum and mean values of the crush force increases when the tube wall or blank thickness increases. Furthermore, the influence of preliminary manufacturing processes on the crash behaviour of automotive body assemblies has been studied numerically by Papadakis et al. [7]. They investigated the effect of manufacturing chain, forming-trimming-welding-crash and found that the energy absorption during crash decreases for models which included structural results due to preliminary manufacturing process steps that resulted from material strain hardening. Wang et al. [8] studied the forming effects on the crash performance of front rail of a typical thin-walled beam structure. Hardening effect from the forming history is found to strengthen the component and significantly influences the crash mode of the front rail in their study. All preceding study mentioned gives profound insight on the importance of including forming process in the crash simulation. Based on the previous study, lack of literature have been found that explicitly explore the effect of forming history based on draw forming process on the simulation of crash analysis. Therefore, this work aims to study the influences of draw forming process in the crash analysis by using a circular cup shape made of advanced high strength steel (AHSS), dual phase steel DP600.

2.0 METHODOLOGY

This work consists of numerical study which includes two different simulation process, forming and crash. Both forming and crash simulations were performed by using dynamic explicit method available in Hyperworks version 13 FE codes using DP600 as working material. The forming simulation has been carried out using single action draw forming incremental approach and for the crash simulations, explicit Radios solver was used.

The forming simulation was performed to obtain the desired shape of a circular cup part as shown in Figure 1 which can be used to represent the automotive production process such as bending (R2 and R6 area), stretching (sidewall area) and compression (flange area).

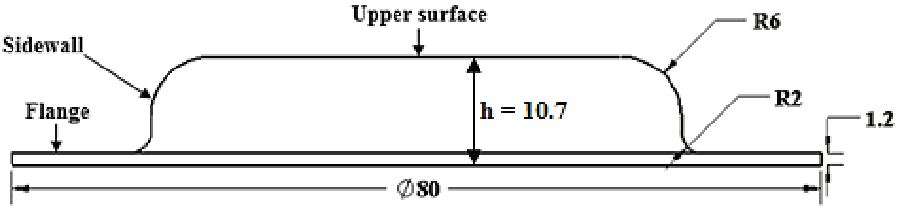


Figure 1: Desired shape of circular cup (All dimensions in mm)

Two type of model was developed from the forming simulation; case 1: without forming histories and case 2: with all forming histories (thickness, stress and strain distribution). The two models were then used to do the crash analysis. For both cases studied, the geometries achieved by the stamping process, e.g., the deformed FE meshes are used as the starting geometry for the crash stage [9]. For case 2 the histories of the forming process were mapped to the circular cup crash model by using mapping process which includes thickness, stress and strain distribution.

3.0 DRAW FORMING SIMULATION

The stamping simulation method used is single action incremental Radioss. The forming tools consist of a sheet metal as deformable steel blank and three rigid parts (punch, die and blank holder) with zero die clearance. Forming tools set-up is shown in Figure 2. A deformable steel blank with 1.2 mm thickness and 85 mm diameter is meshed using 4-node quadrilateral 2D element type. The number of elements and nodes used are 1620 and 1641 respectively. The blank material was made from DP600 and this material has a modulus of elasticity, E = 210GPa, Poisson ratio, v = 0.3 and density, $\varrho = 7800 \text{kg/m}^3$. The meshing strategies performed by using fine mesh at the bending area [10] to capture more realistically the deformation of the circular cup at the curvature of the bending area.

The boundary condition for die was completely constrained while blank holder and punch were allowed for z-axis translation only. Rigid Master, deformable slave surface, stamping – penalty (type 21) contact interface is used for the contact algorithm between rigid parts and deformable blank with a friction coefficient of 0.125.

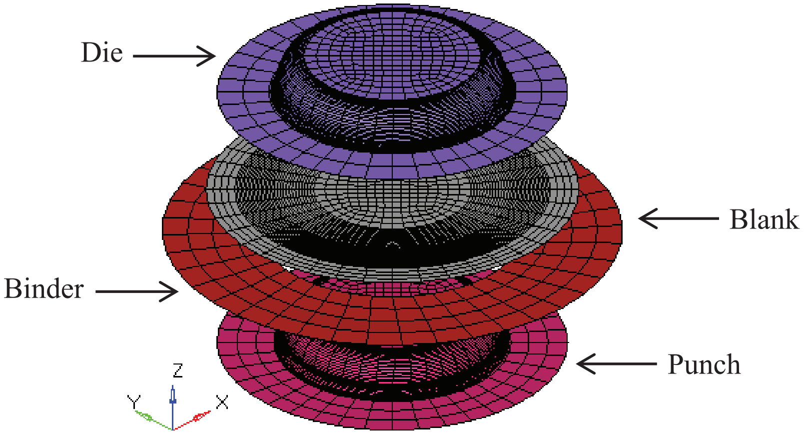


Figure 2: Incremental stamp forming tool set-up

The stamping process was carried out at 50kN binder force (BHF) to form a 9.5mm depth cup which was used to investigate the effect of forming process in crash analysis. The load displacement curve for the forming simulation is shown in Figure 3. The results of plastic strain and thickness change distribution of the stamped part formed as illustrated in Figure 4 was mapped into the crash model for case 2 study.

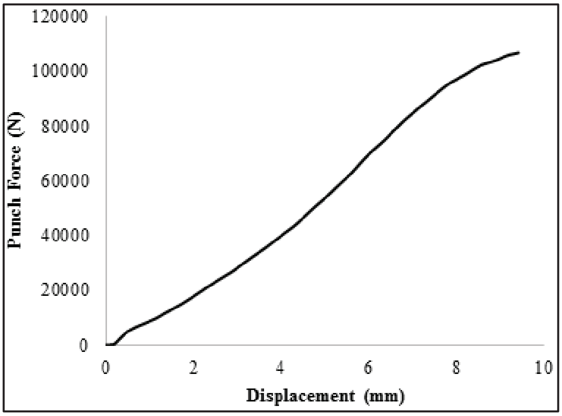
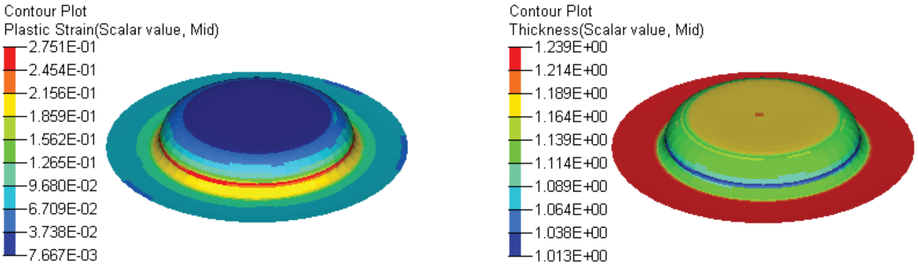


Figure 3: Load-displacement curve of the punch



(a) (b)

Figure 4: Contour plot of (a) plastic strain and (b) thickness change due to forming process for case 2

4.0 CRASH SIMULATION

The crash simulation is defined as a rigid wall body parallel to the x-y plane and approaches towards the circular cup shape parallel to the z-axis. In this simulation, a reference node on the rigid impactor was assigned a constant velocity of 50 km/h or 13.9 m/s to simulate the high impact dynamic loading. The dynamic behaviour of the materials is described using elasto-plastic Johnson-Cook (JC) material model. The material model consists of five constant parameters which are: A, B, n, C and m as shown in Equation (1) where the expression in each bracket provided the strain hardening term, strain rate sensitivity and temperature sensitivity respectively [11]. The JC constants of the circular cup used in the crash simulation are: A = 430 MPa, B = 824 MPa, n = 0.51, C = 0.017 and m = 1.0 [12].

$$\sigma = (A + B\varepsilon^n)(1 + Cln\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0})(1 - T^{*^m}) \qquad (1)$$

where

$$\dot{\varepsilon}_0 = 1/s, \qquad T^{*^m} = \frac{T - T_{room}}{T_{melt} - T_{room}} \qquad (2)$$

The contact between circular cup shape and the rigid impactor is assumed to be frictionless. The modelling and analysis have been performed by using explicit finite element code Radioss. The geometric model is illustrated in Figure 5, which consists of three parts: (1) moving rigid impactor, (2) deformable circular cup, (3) fixed rigid base.

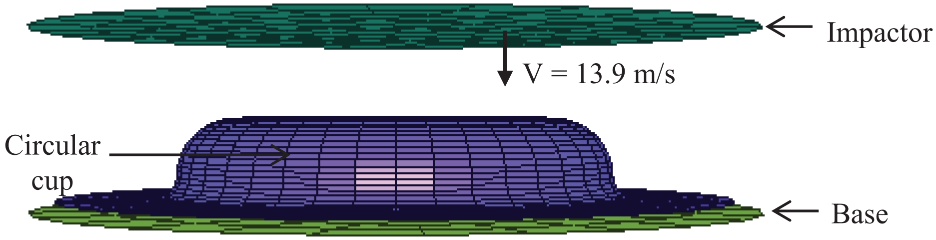


Figure 5: Crash simulation set-up

The circular cup was modelled by using four-node quadrilateral elastoplastic physical hourglass control (QEPH) element. During the crash simulation test, the circular cup was set free to move in all direction and no clamping was applied. The moving rigid impactor and base are modelled using four-node quadrilateral elements and were treated as rigid bodies. The moving rigid impactor was free to translate only along the z-axis and the base was fully constrained.

5.0 MAPPING PROCESS

The effects of forming results on the crash analysis were investigated by performing a mapping process on the crash model. The stamping results were mapped into the crash model by utilizing the state (.sta) file obtained from forming simulations. The state file (geometrical result file) was included to crash model using result mapper. In case 2, the geometrical result of the forming simulation was used as the initial geometry in the crash simulation. Besides that, stress and strain tensors are also transferred and used as the initial stress-strain in the crash simulation. The mapping process is conducted from node to node in order to maintain the reliability of the process.

6.0 RESULTS AND DISCUSSION

The effect of forming results to subsequent crash analysis was studied in terms of its deformation mode, contact force and energy absorption. The results were compared with both cases to investigate the effect of crash analysis with and without forming effects. For both cases, the deformation mode obtained was found to be buckling mode as illustrated in Figure 6. Since the circular cup was impacted without any clamp applied, the upper surface of the cup moves downwards while the flange moves upwards.

Figure 6: Cross-sectional view of deformation mode for both circular cups after impact

The displacement for case 1 is higher than case 2 which is 9.706 mm and 9.462 mm respectively. The result shows that a geometrical model without forming history that has uniform thickness deforms more than the structure that embeds forming history. Without knowing the influence of the manufacturing process on the material properties and thickness distribution, one would arbitrarily choose a high-value of safety factor by uniformly increasing the thickness across the entire structural part in order to be on the safe side [13]. As found from this study, without considering forming history and assuming that the structural part has uniform thickness distribution, the structure deformed more and led to higher energy absorption similar observation was also found by Böttcher et al. [14]. This assumption will lead to inaccurate crashworthiness prediction in the crash analysis since a structural part that is formed from a manufacturing process would change its physical and material properties. The displacement of the circular cup for case 2 is lesser due to work hardening that is experienced by the material during the stamping process. The circular cup part becomes stiffer and stronger when all forming histories are included.

The contact force during the crash analysis also shows a significant difference for both cases studied. The contact force versus displacement is shown in Figure 7. The initial peak load for case 1 is 18700 N. While for case 2 is 33800 N giving 80.7% higher value than case 1. This result shows that forming effects e.g., plastic strain, stress and thickness change from stamping process increases the maximum load increase. The material undergoes work hardening and thickness change during forming process making the structure stiffer to deform. Study conducted by Ryou et al. [15] also found that work hardening experienced by the material during the sheet forming required larger stress to deform compared to the virgin material during collisions.

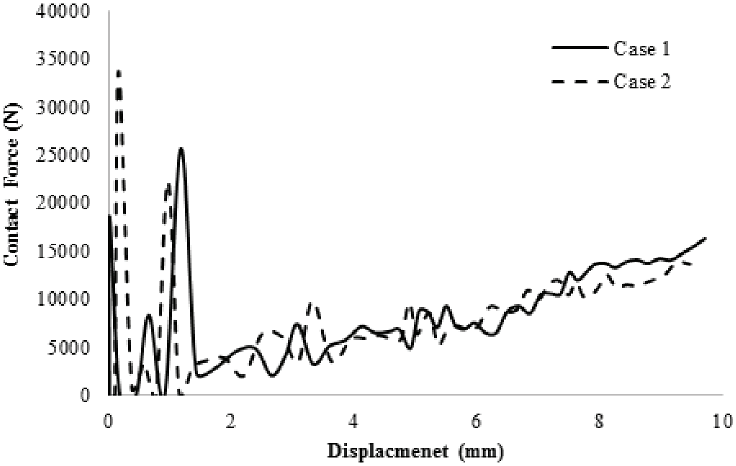


Figure 7: Contact force during crash simulation

The energy absorbed during deformation is plotted in Figure 8. The internal energy of the circular cup for case 1 was higher compared to case 2. The energy absorption increase when the uniform thickness is considered. Because no thickness changes on the model for case 1, the cross sectional thickness of the circular cup was higher and therefore translates into higher energy absorption [16].

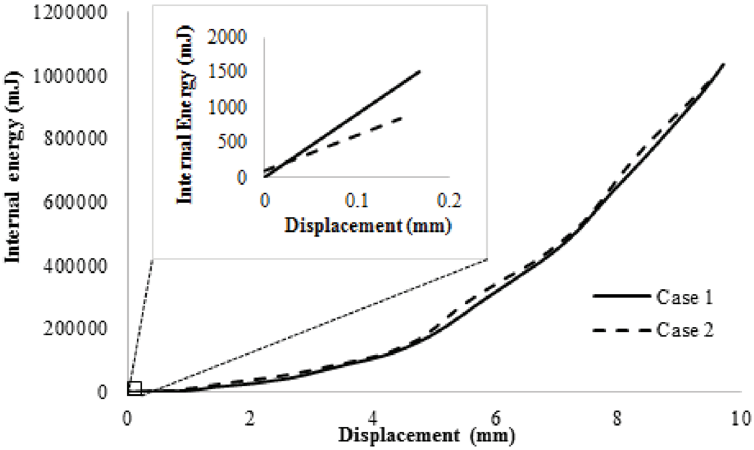


Figure 8: Comparison of energy absorption with respect to the displacement in the circular cup part during the crash event

Case 2 shows incorporating forming results to the crash simulation affect the energy absorption of the circular cup. Initial deformations of forming results are visible in the detailed view of internal energy at which a nonzero value was observed at the beginning of the curve and this result is found to behave as what has been reported by Sasek et al. [17]. This shows that a structural part experience changes in its physical and material properties after undergoing a manufacturing process. The final total internal energy of the circular cup without forming histories is 1040 J and the total internal energy of the circular cup with forming histories is 979 J. The relative difference is equal to 5.87%. This change in energy absorption is a result due to the difference in the initial value of yield strength. As the part undergoes a draw forming process, the final geometry of the product will be formed by plastic deformation that alters the mechanical properties of the product. As a result, the yield strength of the part will be increased due to work hardening. The material strength and thickness affects crash response significantly. The thickness and work hardening used in case 2 give an increase in yield strength for subsequent deformation process. From the results, it is found that the crash analysis of the simple circular cup that was used to represent the manufacturing and design process does show a significant effect on the crash response. Therefore, the crash analysis of a structural part has to be carried out in view of the forming effects for better assessment of crashworthiness to ensure the vehicle and passengers safety during a collision.

7.0 CONCLUSION

Crash analysis of the circular cup with two different initial conditions has been carried out in order to evaluate the crashworthiness assessment for advanced high strength steel, DP600. The difference in the deformation mode, contact force and energy absorption between the results with and without forming histories were investigated.

The comparison explains that the current industrial practice which assumes that a material has a uniform thickness shows different results making the structural part embodying more energy than necessary. It is found that a structural part becomes stiffer when all forming histories are considered in the crash analysis. The contact force increased by 80.7% and the energy absorption reduce by 5.87%. This work shows that draw forming process alters the crash analysis results significantly and therefore forming effects should be considered for more accurate crashworthiness assessment at the design stage of the auto-body part.

ACKNOWLEDGEMENTS

Authors would like to thank Universiti Teknikal Malaysia Melaka for the continuous support on this research project and the financial support from Ministry of Higher Education (MOHE) through research grant no. RAGS/1/2014/TK01/FTK/B00083.

REFERENCES

[1] S. H. Zhang, Z. R. Wang, Z. T. Wang, Y. Xu, and K. B. Chen, "Some new features in the development of metal forming technology," *Journal of Material Processing Technology*, vol. 151, no. 1-3, pp. 39-47, 2004.

[2] A. Senin, Z. M. Nopiah, A. K. A. Mohd. Ihsan, S. Abdullah, D. A. Wahad, M. J. Jamaludin, and A. Zakaria, "Johnson cook constitutive modelling for austenitie metal in hot forming process," *Jurnal Teknologi*, vol. 78, pp. 71-75, 2016.

[3] S.-H. Lee, C.-S. Han, S.-I. Oh, and P. Wriggers, "Comparative crash simulations incorporating the results of sheet forming analyses," *Engineering Computation*, vol. 18, pp. 744-758, 2001.

[4] T. Dutton, S. Iregbu, R. Sturt, A. Kellicut, B. Cowell, and K. Kavikondala, "The effect of forming on the crashworthiness of vehicles with hydroformed frame siderails," *SAE Transactions*, vol. 108, no. 6, pp. 3354-3360, 2000.

[5] D. A. Oliveira, M. J. Worswick, R. Grantab, B. W. Wiliams, and R. Mayer, "Effect of forming process variables on the crashworthiness of aluminum alloy tubes," *International Journal of Impact Engineering*, vol. 32, pp. 826-846, 2006.

[6] A. Najafi, and M. Rais-Rohani, "Sequential coupled process-performance simulation and multi-objective optimization of thin-walled tubes," *Materials & Design*, vol. 41, pp. 89-98,2012.

[7] L. Papadakis, A. Schober, and M. F. Zaeh, “Numerical investigation of the influence of preliminary manufacturing processes on the crash behaviour of automotive body assemblies,” *International Journal of Advance Manufacturing Technology*, vol. 65, pp. 867-880,2013.

[8] W. Wang, X. Sun, and X. Wei, “Integration of the forming effects into vehicle front rail crash simulation,” *International Journal of Crashworthiness*, vol. 21, pp. 9-21, 2016.

[9] A. Govic, R. Moshfegh, and L. Nilsson, “The effects of forming history on sheet metal assembly,” *International Journal of Material Forming*, vol. 7, pp. 305-316,2014.

[10] R. M. Amman, M. F. Halim, D. Sivakumar, I. Abu-Shah, S. N. Sulaiman, and H. Samekto, “Study of thinning effect from deep drawing process on crash analysis,” in *Proceeding of Mechanical Engineering Research Day*, 2016, pp. 37-38.

[11] G. R. Johnson, and W. H. Cook, “A constitutive model and data for metals subjected to large strains, high strain rates and high temperatures,” in *7th International Symposium on Ballistics*, 1983, pp. 541-547.

[12] K. Vedantam, D. Bajaj, N. S. Brar, and S. Hill, “Johnson - Cook strength models for mild and DP 590 steels,” in *AIP Conference Proceeding*, vol. 845, 2006, pp. 775-778.

[13] A. E. Tekkaya, J. M. Allwood, P.F. Bariani, S. Bruschi, J. Cao, S. Gramlich, J. Lueg-althoff, M. Merklein, W. Z. Misiolek, M. Pietrzyk, R. Shivpuri, and J. Yanagimoto, “Metal forming beyond shaping : Predicting and setting product properties,” *CIRP Annals - Manufacturing Technology*, vol. 64, 2015, pp. 629-653.

[14] C. Böttcher, and S. Frik, “Consideration of Manufacturing Effects to Improve Crash Simulation Accuracy,” in *4th European LS-DYNA Users Conference*, 2003, pp. 1-8.

[15] H. Ryou, K. Chung, J.-W. Yoon, C.-S. Han, J. Ryoun Youn, and T. J. Kang, “Incorporation of Sheet-Forming Effects in Crash Simulations Using Ideal Forming Theory and Hybrid Membrane and Shell Method,” *Journal of Manufacturing Science and Engineering*, vol. 127, pp. 182-192, 2005.

[16] N. Abedrabbo, R. Mayer, A. Thompson, C. Salisbury, M. Worswick, and I. Van Riemsdijk, “Crash response of advanced high-strength steel tubes: Experiment and model,” *International Journal of Impact Engineering*, vol. 36, pp. 1044-1057, 2009.

[17] J. Sasek, M. Pasek, K. Benes, and V. Glac, “Effects of Manufacturing Process in Crash Simulations,” *Applied and Computational Mechanics*, vol. 4, pp. 113-120, 2010.