Research on Stamping Forming Simulation and Process Parameter Optimization for a Front Crossbeam of a Car Roof based on FEM

Tayani Tedson Kumwenda[1], Qu Zhoude[2]
Department of Mechanical Engineering,
Tianjin University of technology and education,
Tianjin, 300222, P. R. CHINA

Abstract: **The application of numerical simulation technology based on Finite Element Method (FEM) has continued growing especially in the aerospace and automotive industry. This technological tool has the power to predict common faults such as tearing, wrinkles, springback etc which are common during plastic forming of complex three-dimension automotive body panels. This paper focuses on stamping forming numerical simulation and process parameter optimization for a front crossbeam of a car roof using explicit finite element software DYNAFORM. The effects of three stamping process parameters including blank holder force, drawbead depth and coefficient of friction were studied. The influence of each of these parameters on stamping forming of the front crossbeam of a car roof were evaluated based on orthogonal experiment design method by analyzing the thickness variation of the panel. The final stamping forming simulation results obtained reveals that stamping forming simulation technique based on FEM is essential for material optimization, improved product quality, reduced product development cycle and reduced manufacturing cost. The results also show that orthogonal experimental design method is feasible for parameter optimization in engineering design.**

.

Keywords*: *Sheet metal stamping, numerical simulation, orthogonal design method, Finite Element Method (FEM)

I. INTRODUCTION

Most components that are used in automotive as well as aerospace industry are produced by sheet metal stamping forming process. Sheet metal forming involves a variety of processes amongst which stamping is the most important and its final results directly affect the other subsequent processes. The quality of sheet metal stamped components is affected by various process parameters which many engineers and scholars have continued to research on engineering methods on how to optimize these parameters **[1-4]**. During sheet metal stamping process the material of the sheet metal undergoes plastic deformation as its geometry is transformed into a new desired shape. A major concern during this transformation lies in whether these components which are of complex shape with non-linearity of material that requires high quality and dimensional accuracy can be achieved by the use of traditional design method commonly known as trial-and-error methods. Interestingly, it is surprising to note that despite certain limitations that trial-and-error method exhibit, this method still finds practical application in many industrial fields. However, due to demand for mass production of high quality products having dimensional accuracy over a short period of time, numerical simulation based on FEM in concurrent with various optimization methods has become of great importance in various engineering fields especially in aerospace and automotive and so far surpassed the traditional design methods.

Since the development of FEM technology in the early 1960s, a large number of manufacturing industries including automotive industry have found it prominent to solve complex engineering problems **[5]**. The practical application of FEM numerical simulation based on static explicit and membrane formulation approach on sheet metal forming became realistic towards the late 70's when attempts were made by N.M Wang and B Budiansky **[6]**. Since then various commercial CAE simulation programs have emerged to help tool engineers simulate large deformation of sheet metal structures.

The application of numerical simulation based on FEM in stamping process helps tool designers evaluate the entire process of sheet metal forming thereby predicting common faults such as tearing, wrinkling, springback etc which directly affects the quality of the manufactured components **[7]**. In this paper, several stamping simulation runs have been conducted on a front crossbeam of a car roof using LS-DYNA-based explicit finite element software DYNAFORM to analyze and evaluate the simulation process of the panel during stamping process. Three process parameters including blank holder force F_h, drawbead depth H_1 and coefficient of friction μ_c had been considered for analysis. The influence of each of these parameters on thickness variation of the front crossbeam of a car roof was evaluated and optimized based on orthogonal experiment design method. Through optimal combination values obtained during orthogonal design, the best process parameters which gave optimal numerical simulation results were obtained.

II. FORMING SIMULATION PROCESS

A. FEM Simulation Analysis and Tool Set Up

FEM stamping simulation process involves a number of stages as outlined in Fig.1. In order to obtain precise simulation results for automotive panels, a thorough understanding of each of the stages outlined in the figure is of vital importance.

Generally, complex shaped sheet metal profiles for automotive body panels are generated by CAD software's such as CATIA, Pro/E, UG NX, Solidworks etc and then imported into finite element solvers through standard formats for design optimization studies of die and tooling's. In this research project, a commercial CAD software CATIA was used to generate the front crossbeam of a car roof shown in Fig.2. Geometrical data was then transferred in the form of initial graphics exchange specification (IGES) into finite element simulation software DYNAFORM for simulation analysis.

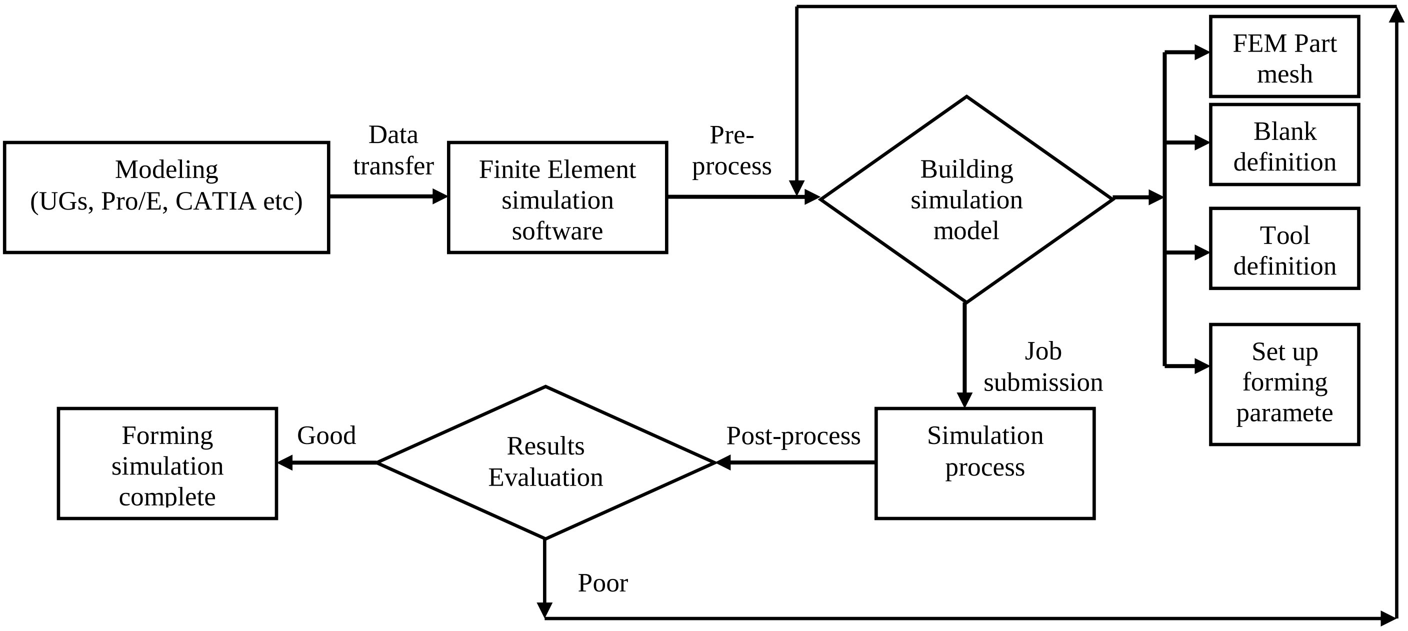

Fig.1-Flow chart of Finite Element Stamping Forming Simulation process

Fig.2-Front Crossbeam of a Car Roof

The panel to be analyzed consists of initial design parameters with overall dimensions of length 930.2975mm, width 138.1028mm and depth 23.5728mm. The panel was constructed using surfaces with literary zero thickness and during actual simulation process 0.7mm thickness was assigned to the blank sheet due to the characteristics of the automotive body panel. As can be seen from Fig.3, the panel also consists of a number of small holes of which the dimensions have been intentionally omitted and in the final simulation model these holes have been eliminated as they have literary no effects on the forming quality of the formed panel.

Creation of the binder, addendum, blank, drawbead and punch for the complete Finite element (FE) simulation model shown in Fig.3 was completed by utilizing eta/DYANFORM inbuilt tools prior to simulation. The meshed blank sheet shown in Fig.4 was obtained by offset tools under preprocess in DYNAFORM of which the boundary line of the part binder was offsetted a suitable distance towards the inner side section.

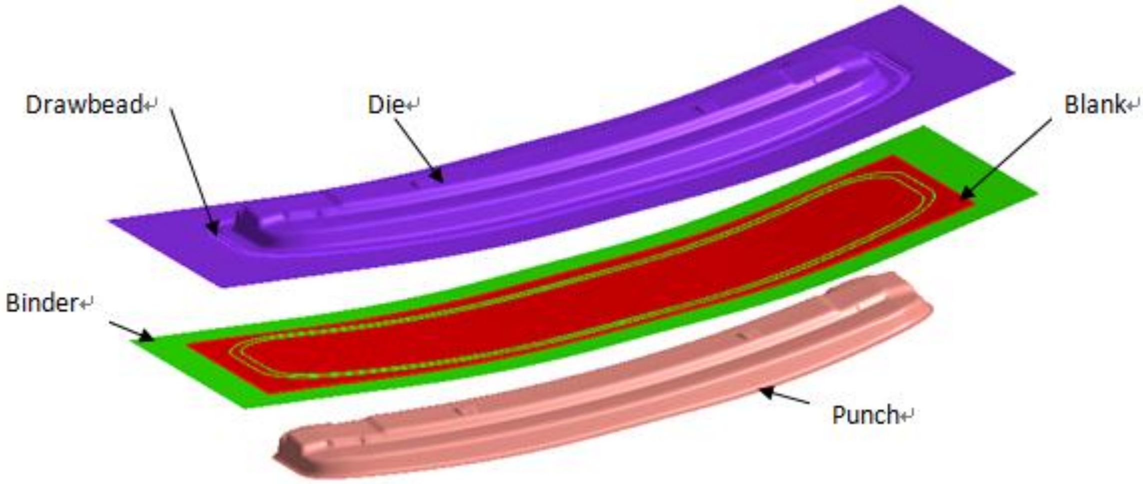

Fig.3-Finite Element Simulation Model

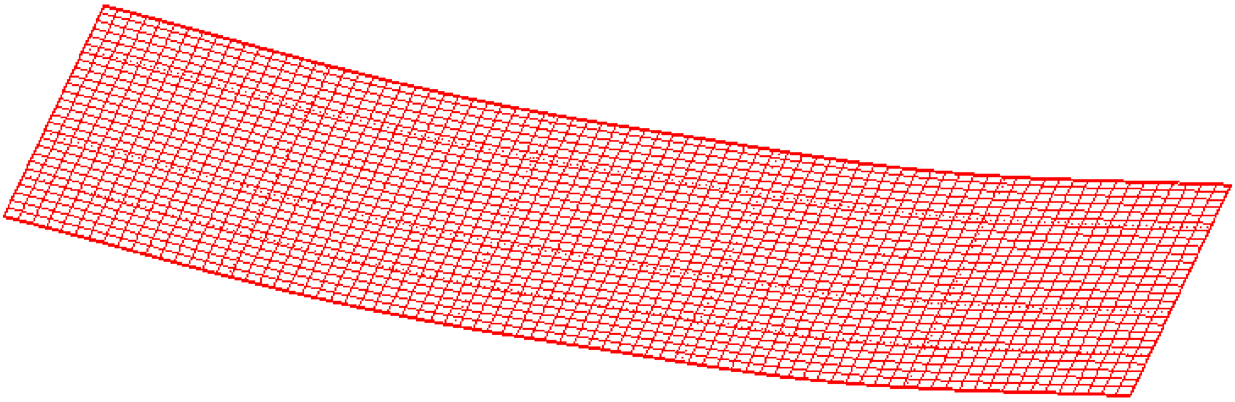

Fig.4-Meshed blank sheet

B. Material Properties and Selection Criteria

In order to obtain precise results during simulation of automotive stamped components, a thorough understanding and selection of the material properties has to be considered. In this research work, steel plate cold drawn (SPCD) material was selected based on its ability to be drawn extensively under cold condition. SPCD is special cold-rolled steel having deep drawing qualities and is basically suitable for automotive inner panels. By utilizing DYNAFORM material library, SPCD material properties were easily obtained as seen in Table 1.

III. PROCESS PARAMETER OPTIMIZATION

Several process parameters have an influence on the forming quality of sheet metal stamping process. In this research paper, three process parameters blank holder force F_h, drawbead depth H_1 and coefficient of friction μ_c had been

Table 1: Mechanical Properties of SPCD

Mass Density ' ρ ' (kg/mm^{-3})	Young Modulus 'E'	Poisson's Ratio ' ν '	Yield Strength ' σ_Y ' (N/mm^2)	Thickness (mm)	Strain-hardening Exponent 'n'	R_{00}	R_{45}	R_{90}
7.85x10^{-9}	2.07x10^5	2.8x10^{-1}	240	0.7mm	0.233	1.85	1.37	2.02

considered for evaluation and optimization purposes of the stamping forming process for the front crossbeam of a car roof. In order to insure optimal results, orthogonal experiment design method was adopted for process parameter optimization. This engineering method employs an orthogonal matrix array to search for optimal combination values ensuring that fewer experiments or numerical simulation runs are conducted during the forming process. Through analysis of the evaluation target, the process parameters can easily be optimized and the maximum thickening rate as well as maximum thinning rate can easily be evaluated. The initial blank holder force F_h for numerical simulation is determined based on calculations using the following formula;

$$F_h = Aq \qquad (1)$$

Where $\boldsymbol{A}$ represents the area of contact between the blank holder and the blank sheet in mm^2 , $\boldsymbol{q}$ is the blank holder force per unit area in ***MPa*** which is a constant and is often dependent on the material thickness of the part being formed. For steel greater than 0.5mm thick the value of $\boldsymbol{q}$ varies from 2.0 ~ 2.5MPa and in this case study 2.4MPa was considered for calculations. The blank holding force was hence calculated and found to be approximately 555KN . Since stamping forming simulation of a front crossbeam for a car roof involves multiple influencing factors, two experiment values for blank holder force i.e. 500KN , and 600KN were chosen for running the simulations.

In order to account for thickening during forming process, tool clearance between the die and the punch was left at default 1.1t which is slightly higher than the thickness of the blank sheet. Coefficient of friction for this study was randomly set to 0.10 and 0.125 respectively with assumptions that the smaller the value the better the forming quality during process.

To further control the flow of material during the process, a circular drawbead type was chosen and its structural profile was generated by offsetting the boundary line of the part binder a distance of 25mm from the inner edge extending outwards. The drawbead line was hence locked to the die part and at the same time the male of drawbead was set to be at the die with the female portion being on the binder. The depth $\boldsymbol{H_1}$ of the drawbead illustrated in Fig.5 an extract from eta/DYNAFORM 5.9 user manual **[8]** was set to 5mm and 7mm respectively. The amount of drawbead restraining force was set to a fixed value of 120N/mm while other parameters shown below were based on default as from eta/DYNAFORM.

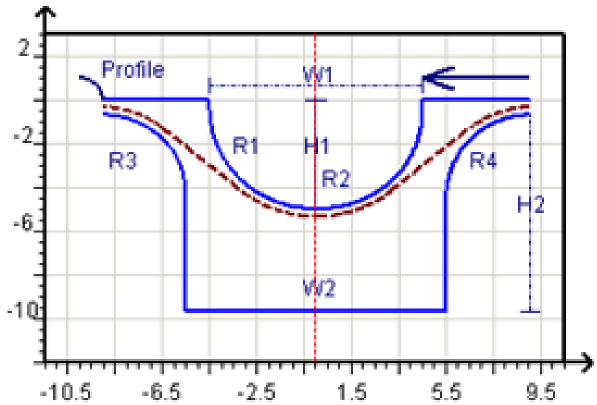

Fig.5-Drawbead profile

Table 2: orthogonal experiment design showing factors and levels setting

Levels	Factors (process parameters)		
	Blank holder force F_h	Drawbead depth H_1	Coefficient of friction μ_C
1	500KN	5 mm	5 mm
2	600KN	7 mm	7 mm

Following the above analysis, an orthogonal experiment design table showing three factors and two levels displayed in table 2 above was established. Based on the orthogonal experiment design table, one can notice that a total of $2^3 = 8$ simulation runs were to be conducted in order to obtain the entire experiment results. However, the number of simulations had been minimized by orthogonal array of the order L_4 (2^3) as represented in the orthogonal experiment scheme table 3. Similar to the previous table, the designed orthogonal experiment factors i.e. blank holder force, drawbead depth and coefficient of friction have been represented by special symbols F_h , H_1 and μ_c respectively.

Table 3: Scheme of Orthogonal Array $L_4\,(2^3)$

Levels	Experiment No.	Influencing Factors		
		F_h	H_1	μ_c
1	1	500	5	0.10
	2	500	7	0.125
2	3	600	5	0.125
	4	600	7	0.10

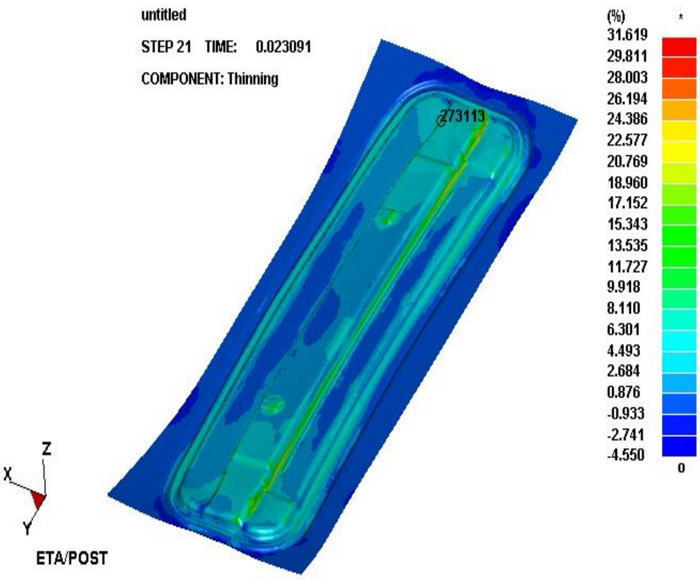

Fig.6-Thinning distribution for experiment 2

IV. EXPERIMENT RESULTS AND DISCUSSION

As stated in the previous section, four (4) numerical simulations were carried out in this analysis and the results corresponding to the maximum thickening rate as well as maximum thinning rate in terms of percentage of the original blank sheet thickness of the panel are displayed in table 4 for analysis purposes. Based on the numerical simulation results, it is clear to note that almost 75% of the panel falls within the safe zone with values for the maximum thinning rate ranging between 20%□30% which is in agreement with the average acceptable range. However, maximum thinning rate tend to have exceed greatly at level 1 experiment 2 which is likely to put the front crossbeam being analyzed at high risk of tearing which may lead to failure of the panel. The maximum thinning rate is quite significant at node ***273113*** located at the far end along the edge of the panel. Minor wrinkles also seem to have been observed on the binder portion with some parts exhibiting great signs of insufficient stretching. Fig. 6 and 7 shows thinning distribution and the forming limit diagram for experiment 2 of level 1 for the maximum thinning rate.

On the contrary, numerical predication for maximum thickening rate shows that all experiment results are of acceptable standard as can be seen in table 4. The maximum thickening rate is in accordance with the general rules of stamping forming which should is 5% of the original thickness or below.

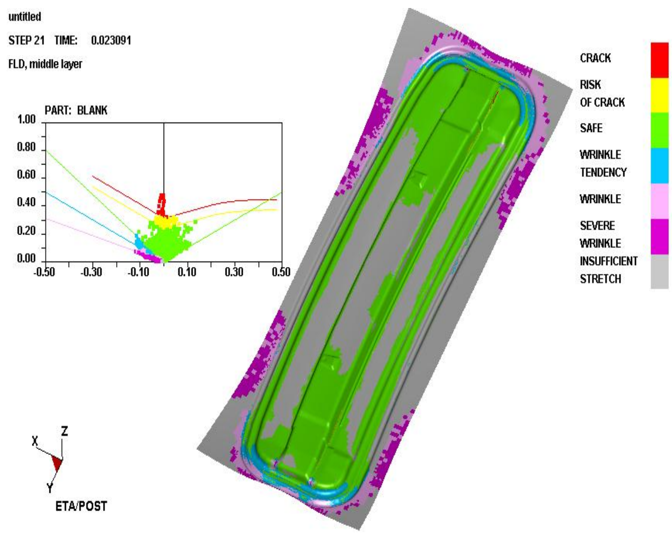

Fig.7-Forming Limit diagram of the panel for experiment 2

Table 4: Thickness and Thinness Simulation results based on orthogonal experiment design

Levels	Experiment No.	Influencing Factors			Simulation results	
		F_h	H_1	μ_c	Maximum thickening rate (%)	Maximum thinning rate (%)
1	1	500	5	0.10	4.43	26.90
	2	500	7	0.125	4.57	31.62
2	3	600	5	0.125	4.00	30.84
	4	600	7	0.10	4.86	29.60

On the other hand, the average combination for the thickening and thinning rates of the panel at two different levels based on the results obtained in table 4 are shown in table 5. In this table, K_1 and K_2 corresponds to the total sum of the maximum thickening and maximum thinning rates for all the two designed levels with respect to the selected influencing factors. The mean averages for the two levels are designated by $\bar{K}_1$ and $\bar{K}_2$ respectively. The degree at which each selected factor influence the formability of the panel is represented by R. The value of R is calculated by subtracting the smallest mean average from the greatest mean average. The results in table 5 reveals that the best optimum levels for these process parameters which can give us acceptable maximum thickening rate are corresponding to $F_h = 600KN$, $H_1 = 5mm$ and $\mu_C = 0.125$ with drawbead depth being the greatest influencing factor as can be seen against the order of influence in table 5. On the contrary, the maximum thinning rates' optimum levels combination average is seen to be corresponding to $F_h = 500KN$, $H_1 = 5mm$ and $\mu_C = 0.10$ with process parameters exhibiting a different order of influence from that of maximum thickening rates which $H_1 \rightarrow \mu_C \rightarrow F_h$. The graphs in Fig.8 and Fig.9 illustrate the magnitude at which each combination average influence the forming quality of the body panel at two different levels.

Table 5: Thickness and thinness combination average at two different levels

Combination average	Maximum thickening rate (%)			Maximum thinning rate (%)		
	F_h	H_1	μ_c	F_h	H_1	μ_c
K_1	9.00	8.43	9.29	58.52	57.74	56.50
K_2	8.86	9.43	8.57	60.44	61.22	62.46
$\bar{K}_1$	4.50	4.22	4.65	29.26	28.87	28.25
$\bar{K}_2$	4.43	4.72	4.29	30.22	30.61	31.23
R	0.07	0.50	0.36	0.96	1.74	2.98
Optimum combination levels	*600*	*5*	*0.125*	*500*	*5*	*0.10*
Order of influence	$H_1 \rightarrow \mu_c \rightarrow F_h$			$\mu_c \rightarrow H_1 \rightarrow F_h$		

Comparing the two graphs, one can notice that the behavior of thickness variation for both maximum thickening rate and maximum thinning rate for the panel tend to be in an opposite trend. In Fig.8, the maximum thickening rate decreases rapidly as the value of coefficient of friction increases between the two levels. On the contrary, there is a minimal decrease in terms of thickness as the amount of blank holder force increases from 500KN to 600KN. Apart from that, we can also notice that there is shape increase in thickness as the drawbead depth increases. On the other hand, results for all the three influencing factors shown in Fig.9 tend to be increasing the amount of maximum thinning rate of the panel with respect to their factor levels. The graph in Fig.9 reveals that coefficient of friction has the greatest influence on maximum thinning rate.

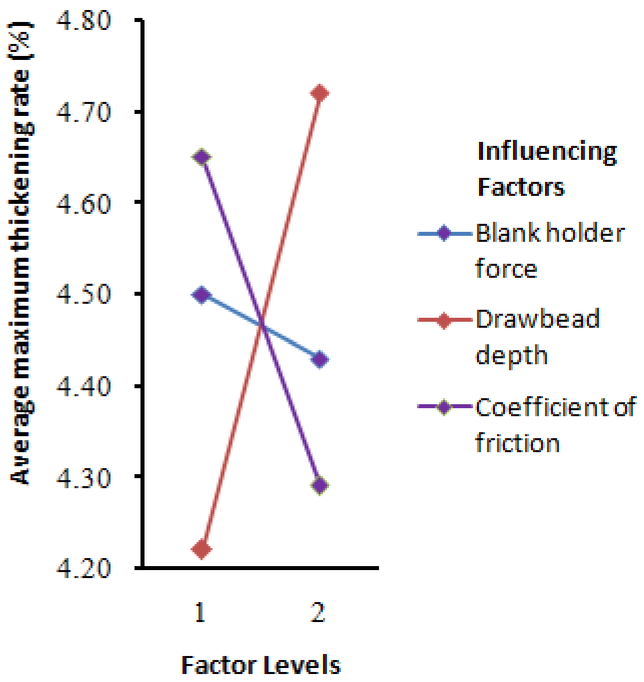

Fig. 8-Maximum thickening level average

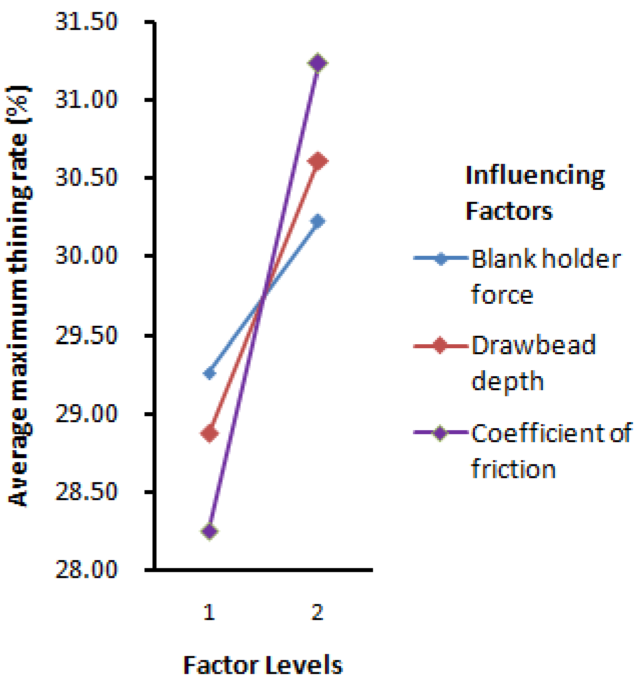

Fig. 9- Maximum thinning level average

In order to arrive at the best optimized process parameters, the maximum thickening rate and maximum thinning rate optimum combination levels obtained in table 5 were analyzed by comparing the actual numerical simulation results obtained earlier during the earlier simulations. The forming limit diagrams illustrated in Fig.10 and Fig.11 illustrates numerical simulation results for the best optimum levels for maximum thickening rate and maximum thinning rate obtained by eta/DYNAFORM. From Fig.10, one can notice that the optimum combination levels for maximum thickening rate are not meeting the requirements as the panel displays high levels of tearing which can be depicted by red on the forming limit diagram. The maximum thinning rate distribution for this simulation model also reveals that the value for maximum thinning rate has exceeded the acceptable limit and is not suitable for production as it can easily fail. On the other hand, the forming limit diagram obtained from simulation of the maximum thinning rate optimum combination shows good results with most parts of the panel in the safe zone as can be seen in Fig.11. Therefore, the best optimized parameters were chosen to be $F_h = 500KN$, $H_1 = 5mm$ and $\mu_C = 0.10$.

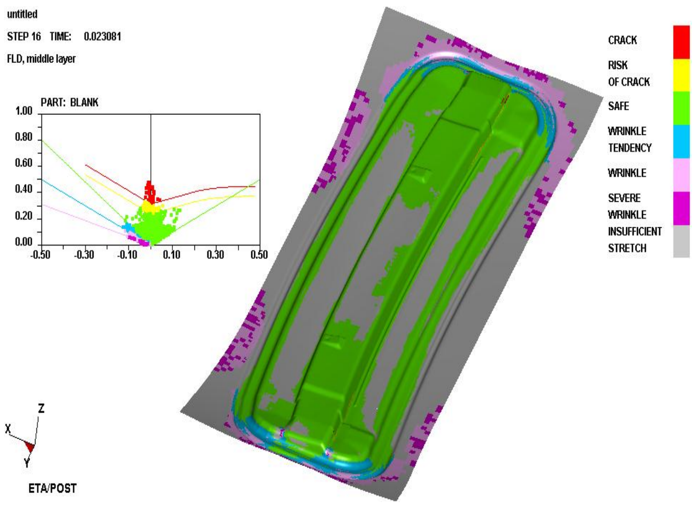

Fig. 10-Forming Limit diagram for the maximum thickening rate optimum levels

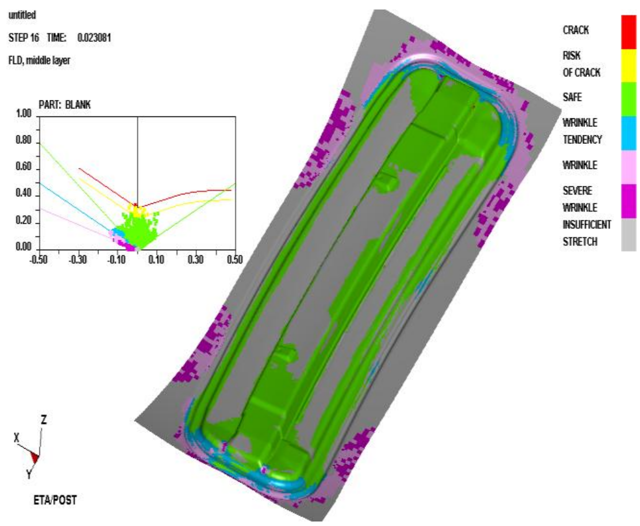

Fig. 11-Forming Limit diagram for the maximum thinning rate optimum levels

V. CONCLUSION

In this research work, the influence of three process parameters blank holder force F_h, drawbead depth H_1 and coefficient of friction μ_c were analyzed and evaluated based on orthogonal experiment design method. Through this research, the following conclusions were summarized;

- The greatest process parameter influencing the quality of stamping forming simulation of the front crossbeam of a car roof obtained by evaluating the maximum thickening rate and maximum thinning rate is not always the same.
- The influence of blank holder force, drawbead depth and coefficient of friction were effectively optimized based on orthogonal design method, $F_h = 500KN$ $H_1 = 5mm$ and $\mu_c = 0.10$ were found to be the best optimal process parameters that can give acceptable forming results.
- Orthogonal design method in concurrent with FE numerical simulation can be used to quickly arrive at optimum results thereby serving a lot of time and money in manufacturing engineering.
- Numerical simulation based on Finite element method (FEM) can be used for analyzing complex three dimension automotive body panels.
- Further research has to be done in order to improve the efficiency of finite element simulation software's in order to minimize the time for simulation process.

REFERENCES

[1] Ma, Guoying; Huang, Binbing. (2014). Optimization of Process Parameters of Stamping Forming of the Automotive Lower Floor Board, *Journal of Applied Mathematics*, 2014: pp. 1-10.

[2] Lee, Jae-Jun; Park, Gyung-Jin. (2014). Optimization of the Structural and Process Parameters in the Sheet Metal Forming Process, *Journal of Mechanical Science and Technology*, 28(2): pp. 605-619

[3] Zhang, Jinya; Zhu, Hongwu; Li, Yan; Yang, Chun. (2010). Optimization of Helico-Axial Multiphase Pump Impeller Based on Orthogonal Experimental Design, *Fourth International Conference on Experimental Mechanics*, Proc. SPIE 7522:pp. 1-8

[4] Yan, Huajun; Zhang, Shuangjie; Liu, Yuzhong (2015). Parameter Optimization for Aluminum Alloy Magnet Cover With Stamping Based on FEM Simulation, *3rd International Conference on Material, Mechanical and Manufacturing Engineering*, (IC3ME 2015): pp. 2079-2083

[5] Ray W. Clough. (1990). Original Formulation of the Finite Element Method, *Finite Element in Analysis and Design*, 7(2): pp.89-101. Wang, N. M.; Budiansky, B. (1978) Analysis of Sheet Metal Stamping by Finite-Element Method, *Journal of Applied Mechanics*. 45(1): pp. 73-82

[6] Tisza, M. (2004) Numerical Modeling andSimulation in Sheet Metal Forming, *Journal of Materials Processing Technology*, 151 (1-3): pp. 58-62

[7] Engineering Technology Associates inc. (1998-2009). eta/DYNAFORMTM 5.9 Application Manual. Troy: Engineering Technology Associates

[8] Wang X F, Lang L H. (2010). Sheet Metal Forming CAE Design And Application-Based on DYNAFORM [M]. Beijing: Beijing University of Aeronautics and Astronautics Press (Chinese)