ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

IARJSET

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

Sheet Metal Forming Simulation of Draw Panel Using FEM

A.S.Nasikkar[1], Prof. V.S.Aher [2]

Department of Mechanical Engineering, Savitribai Phule Pune University, Pune, Maharashtra, India[1]

PG Coordinator ,Department of Mechanical Engineering, Savitribai Phule Pune University, Pune, Maharashtra, India[2]

Abstract: In Automobile industry sheet metal forming process are getting more complicated due to complex part geometries. In this Paper we have used simulation software to simulate the forming process before actually manufacturing of the tools. For this purpose commercial FEM simulation software autoform was used to simulate the draw component panel under study . The main objective was to avoid cracking and severe wrinkling which may result in the forming process..This defects occurring in sheet metal forming are bound to occur in the draw component . Conventionally these defects are reduced by varying the forming process conditions trial and error method. This trial and error method causes loss in terms of money and time which finally increases product development time . There are various parameters included in the forming process which affect the final products quality. The most effective process parameters are identified using FMEA and this are blank holder force, die entry radius and draw bead height . Combination of different values of these process parameters is done using Design of experiments (DOE) by Taguchi's orthogonal arrays in Minitab software. Thus trial and error methods are replaced by the virtual simulations of these trials using Finite Element Method (FEM) based software and optimization is carried out by using autoform software. This method will replace the need of industrial expertise and also save a lot of cost and time. The results of optimization are validated by actual formed component at industry using same optimized parameters. With help of simulations a stable forming process which did not yield cracks or severe wrinkling, was eventually found.
.

Keywords: Sheet metal forming, Thinning, Finite element method, Design of experiments (DOE), Taguchi, Minitab

I. INTRODUCTION

Sheet metal forming is widely used in most industries for forming sheet into appropriate shape by plastically deforming the sheet metal beyond its yield strength to achieve permanent deformation. The major application of sheet metal is in automobile industry. Which include door, fender, dumper, roof panel and seat frame. An automobile industry is growing rapidly, the demand for precise and accurate information concerning part design and formability of metal sheet becomes essential [1]. Strong understanding of forming process is critical to produce high quality and cost effective products. Hence most of the automotive industry uses sheet metal forming simulations during the vehicle development process in order to accelerate the design cycles and to reduce development costs. The simulations are applied to assess the feasibility of part geometries during the product design phase, to try out prototype and production tooling during the die development process, and to optimize process parameters for maximum efficiency, reliability and quality[2]. However, the defects of a work piece occur due to wrinkling, spring back, material failure and others . The problems of the defects can be improved by optimization of the metal forming process. Two kinds of design variable groups exist in this optimization; one is the structural parameter group and the other is the process parameter group. The structural parameters are the initial size, shape of the blank, etc. which are the geometries of the work piece. The process parameters are the working conditions such as the punch velocity, the blank holding force , the draw bead length , the friction factor, etc. A different optimization method is employed for each group of the parameters. Researches on the optimization of structural parameters are typically based on an interpolation method an inverse finite element method and others. The research goal using these methods is to determine the initial blank shape for the desired final shape. When the plastic deformation path of metal and other parameters are not considered exactly, these researches have the disadvantages of increasing errors and overlooking material failure. Optimization of the process parameters is employed when the wrinkling and spring back phenomena should be improved and material failure is considered [3]. Therefore, it is important to optimize the process parameters to avoid defects in the parts and to minimize production cost. Optimization of the process parameters such as die radius, blank holder force, friction coefficient, etc., can be accomplished based on their degree of importance on the sheet metal forming characteristics. The objective of the work is to successfully simulate the forming operation of a sheet metal component and validate the results of simulation by actual trials. For this analysis work we use finite elements method (FEM). The FE analysis software is regularly employed in the design assessment of stamping tooling and dies in automotive industries, and the process simulation approach has been established as a practical methodology in the part formability and stamping failure. Develop the die design and establish process simulation in metal forming by, Predicting metal flow and final dimensions of the formed

DOI 10.17148/IARJSET/ICAME.48

part Preventing flow induced defects such as excessive thinning and wrinkling, predicting limit strains, strain history and effect of different properties on formability of the metal Improve part quality and control of geometrical complexity, while reducing. Manufacturing cost by, Reducing die try-outs and lead times, Reduce rejection and improving material yield .More and more industries are utilizing finite element analysis (FEA) techniques to simulate various material forming processes. In many cases, results from the finite element analysis provide sufficient information to prevent potential defects and develop solutions to correct anticipated problems, which may occur during the actual sheet metal forming.

II. SHEET METAL FORMING SIMULATION

The aim of most current sheet metal forming research is to minimize the time and cost for process development and production while minimizing scrap and optimizing the quality of the parts produced. Thus numerical simulation of sheet metal forming process is very useful tool for analysis. Finite element analysis is one of the methods recognized both by researchers as well as industrial practitioners to be the key enabling technology for achieving these goals over the last two decades. The rapid development of finite element (FE) technology has made significant contribution to the sheet metal forming industry. During the product development cycle, FE analysis codes are playing an important role from quick qualitative check at the conceptual level, followed by quantitative validation at tryout phase up to the final process validation and tuning. Fig. 1 shows the sheet metal forming simulation process [4] ,[5].

A. Steps Involve In Sheet Metal Forming Simulation

Finite element analysis application to sheet metal forming follows typical procedure. There are many software available for simulation of forming processes. Fig. 1 shows flow chart for stamping simulation. Some of the basic steps involved in stamping simulation are as follows :

- Importing CAD surface model of designed tools
- Meshing of initial tools
- Filleting of tools after meshing
- Meshing and importing of draw bead
- Creation & mesh generation of the blank
- Process set up (carrying out tool assembly)
- Run the solver to carry out simulation [6]

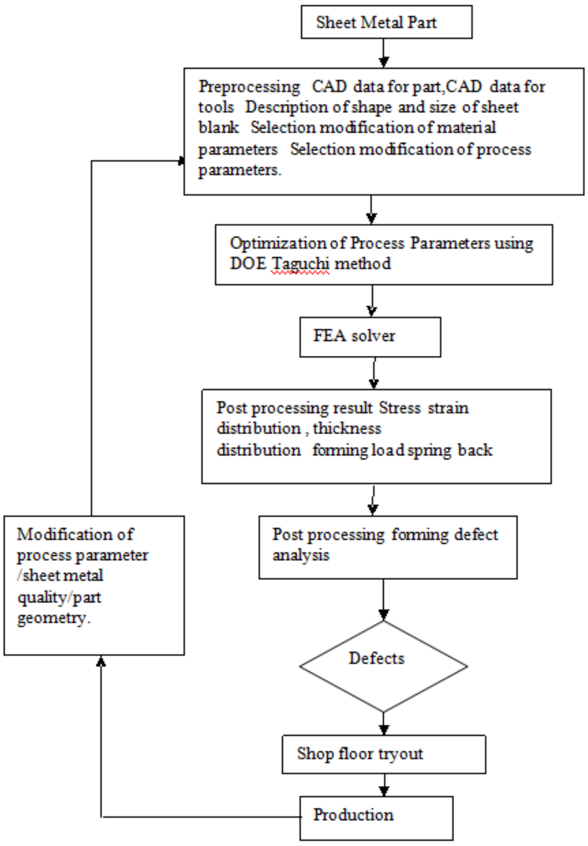


Fig.1. Flow chart showing sheet metal forming simulation process .

B. Problem Defination

The part under study is a draw panel part of an automobile. So the component must have sufficient strength, good accuracy. While manufacturing of the component there is risk of defect of wrinkles, thinning, splits, and spring back on the draw panel To overcome this defect we have to optimize the process parameters to eliminate defect and reduce tryout time. Optimization of process parameters such as die radius, blank holder force, friction coefficient, punch entry radius, draw bead position and bead radius, this can be done by using sheet metal forming simulation software. To find the major cause of defect we have done FMEA of the draw panel.

III. METHODOLOGY

A. Ishikawa Diagram

The cause and effect diagram also known as Ishikawa diagram is used to find problems in the drawing process. The improvement group developed a diagram with brainstorming session conducted. As shown in Fig 2 the starting point of the cause and effect diagram was the question In the X-Y model, Y corresponds to the number of complaints and X to the causes to these complaints. The improvement group was able to find the important root cause to the problem. These causes were chosen, since they were detected frequently and will work as input to the process FMEA [5] .

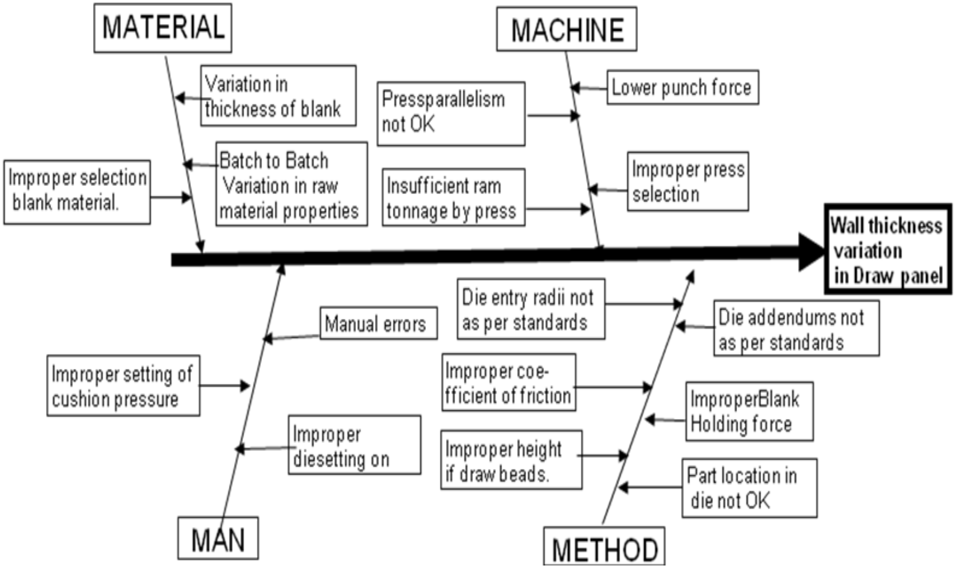


Fig 2. Ishikawa Diagram for draw process

B. FMEA Of Draw Component

After Ishikawa we use failure mode and effect analysis (FMEA) technique to identify critical process parameter for drawing of component. FMEA is one potential tool with extended use in complicated assemblies aerospace and automotive industries. The main purpose is to reveal system weaknesses and there by minimize the risk of failure occurrence. One of the most important quality-management techniques is FMEA. It is devoted to minimizing the risks of failure and understanding what actions need to be taken as a result of significant unplanned events. In our work we use design FMEA to find the highest RPN value for defect and change process parameters to achieve optimum solution to problem. The FMEA results are then presented in an appropriate table, together with the RPN and the three important indicators of failure severity, occurrence, and detect ability. The RPN is calculated as the product of the three Characteristic failure indicators: quality field but also in the occupational health/safety and environmental sector, which is of great interesting the insurance industry (risk analysis of industrial accidents with major environmental and/or human impact). Failures allocated to the method affected by human factors.

RPN = (Severity) × (Occurrence) × (Detect ability) (Eq 1) for every single failure mode, the corresponding columns concerning potential effect of failure and po tential cause mechanism of failure are completed. In order to quantify the failure risk (RPN), it is necessary y to evaluate the principal FMEA indicators: severity (effects of failure), occurrence (frequency of failure cause), and detect ability (process controls).

These indicators are estimated by using a statistical analysis of the process where severity is the failure criticality indicator and is graded within the range of 1 to 10 (1, low criticality; 10, high criticality); occurrence is the failure frequency indicator and is graded within the range of 1 to 10 (1, low frequency; 10, high frequency); and detect ability is the failure detection capability and is graded within the range of 1 to 10 with decreasing capability (1, high detection capability; 10, low detection capability).

TABLE I: FMEA OF DRAW COMPONENT

Process: Draw operation		
Potential Failure Mode	**Potential Effect of Failure**	**Potential Cause of Failure**
Draw panel Defects	1)Fitment problem during assembly of component	1.Insufficient ram tonnage by press
	2)Incomplete part profile	2.Die entry radii not as per standards
	3)Aesthetic rejection at the customer	3.Press parallelism not OK
	4) Chance of Tear part at forming position	4.Improper press selection
		5.Lower punch force
		6.Manual errors
		7.Improper die setting on press
		8.Improper coefficient of friction
		9.Improper setting of cushion pressure
		10.Improper selection of blank material.
		11.Batch to Batch Variation in raw material properties
		12.Improper coefficient of friction
		13.Improper height if draw beads.
		14.Part location in die not OK
		15.Improper Blank Holding force
		16.Die addendums not as per standards

TABLE III: FMEA OF DRAW COMPONENT

Process: Draw operation				
Design Control Prevention	**S**	**O**	**D**	**RPN**
1.Provide tonnage Indicator	6	2	2	24
2.Use correct value of die entry radius from simulation	6	6	8	288
3.Check press parallelism	2	4	2	16
4.Calculate draw tonnage and check press capacity	2	2	4	16
5.Machine setting not ok	4	4	4	64
6.Use sensor for part loading	4	2	4	32
7.Use skilled operator	4	2	4	32
8.Use optimum coefficient of friction	4	4	2	32
9.Adjust cushion pressure	2	4	6	48
10.Selection of deep draw quality of material	4	6	2	48
11.Check material by test lab certification	2	4	4	32
12.Use proper lubrication	6	4	6	144
13.Use optimum draw bead high	4	6	10	240
14.Check nesting of part	6	4	4	96
15.Optimize blank holding force	6	8	6	288
16.Use design check sheet	4	4	4	64

The factors for draw panel evaluation consist of a series of criteria used to evaluate the risk priority of a component. In this evaluation two major defects are considered wrinkles, crack. The classification criteria of each one of those parameters are presented in Table I from this parameters , it is defined the called RPN which is calculated by product of three previous indices (Severity) × (Occurrence) × (Detect ability) and the result are shown in Table II . The blank holding force ,draw bead hight and die entry radius are the most important component . Increase or decrease in this parameters result in crack or wrinkles in draw panel , hence the optimum value of process parameters must be taken for trial purpose.

C. Selection of Orthogonal Array

Techniques of laying out experiments under multiple factors had been known for long time and are known as the Factorial DOE. This method helps the researchers in determination of the possible combinations of factors. However in industrial settings it is extremely costly process to run large number of experiments in testing all combinations. The Taguchi approach delineate the rules in carrying out experiments and are further simplified and also standardized the design of the experiment along with minimum number of factor combinations that would be required for testing the influence of diverse factors [13] .

D. Experimental study on Drawing

In this study, an experimental method is proposed to fully understand effects of various parameters on thickness variation of cup formed by drawing process. The drawing tests were conducted on 500T mechanical press with the tool setup as shown in Figure 4 and the corresponding tool specifications for blank, punch, die, blank holder force and draw bead height used in deep drawing process are as shown in Table III. Among the various factors influencing the deep drawing process, blank holder force, die shoulder radius and bead height play an important role in quality of the formed part and hence, blank holder force, die shoulder radius and bead height are considered in the optimization of deep drawing process as shown in Table III . L9 orthogonal array had used to investigate the effect of bead height, die shoulder radius and blank holder force on thickness variation by conducting only nine experiments under three levels of each parameter[13] .

TABLE IIIII:ORTHOGONAL ARRAY L9 OF TAGUCHI METHOD

Experiment No.	**Parameters**		
	Draw Bead Height	**Die shoulder radius**	**Blank holder force**
	Dbh	Rd	F_h
1	1	1	1
2	1	2	2
3	1	3	3
4	2	1	2
5	2	2	3
6	2	3	1
7	3	1	3
8	3	2	1
9	3	3	2

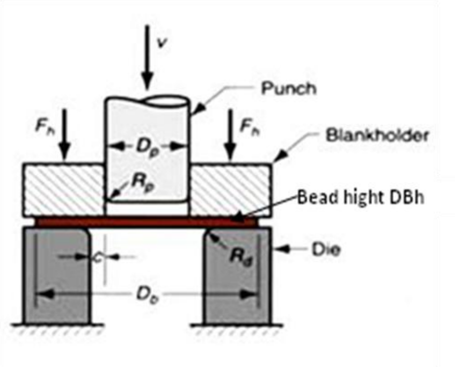


Fig. 3 Scheme of deep drawing process

E. Formability Analysis of Draw Panel

The formability of blank sheet depends on the process parameters such as pressure, punch speed, friction coefficient, and blank holder force. Crack and wrinkle are the major modes of failure in sheet metal parts. Hence, using proper process parameters are essential to restrict wrinkling tendency and avoid tearing. One of the quality criterions in sheet metal formed parts is thickness distribution In order to evaluate the possibility of wrinkling, cracking etc. the strains in the formed component are analysed and compared against the forming limit curve, Fig.4. This curve is extracted from biaxial strain tests, for example via the Erichsen test. The test specimen of the material has been drawn until fracture or diffuse necking. The curve that forms the lower boundary of the area C is the forming limit curve. The curve describes the level of strain that the actual material can withstand until failure, cracking or wrinkling occurs. Following a rule of thumb experience to assure that the component not will break the strain level should not exceed 80% of the level of the forming limit curve.

The different areas in the diagram are:

- Zone A is Recommended appropriate use of the forming abilities of the material
- Zone B Danger of rupture or cracking.
- Zone C The material has cracked.
- Zone D Severe thinning.
- Zone E Insufficient plastic strain, risk of spring back
- Zone F Tendency to wrinkling.
- Zone G Fully developed wrinkles.

On the FLD, the forming limit curve (FLC) indicates the forming limits of the material. It divides the diagram into two zones Safe zone: The area in which failure will not occur during forming. Failure Zone:The area in which the material may exhibit localized thinning. Failure is defined as the appearance of localized thinning or necking, not necessarily fracture.

TABLE IVV:MATERIAL PROPERTIES

Property	Value.
Work hardening index (n):	0.241
Yield strength	165 MPa
Lankford Coefficient (r)	1.8
Young‘s Modulus	210,000
Strength coefficient	501 MPa
Poisson ratio	0.3

F. Mechanical properties of component under study

In this study, a draw panel with EDD steel and blank thickness of 0.8 mm is simulated by using Auto form to study the effect of these parameters on failure modes and thickness distribution. The parameters for simulation are shown in Table IV.

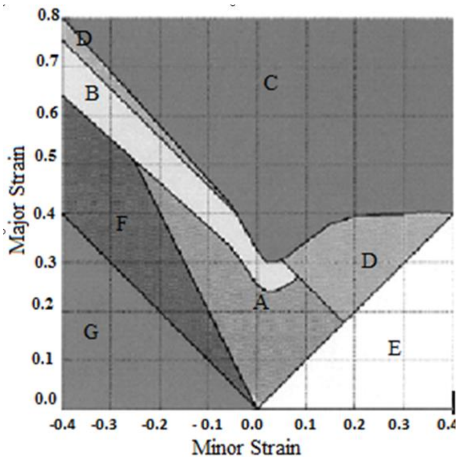


Fig.4 Forming limit diagram of major vs. Minor strain, distinguishing different dominion [4]

F. Process parameters

To successfully form the component several iterations were performed by varying blank holder force from 55 ton to 65 ton we have carried out simulation trials at every 5 ton increment of blank holder force the remaining parameters are kept constant. The value of process parameters for simulation trial is shown in Table V

TABLE V: VALUE OF PROCESS PARAMETERS

Description of parameter	**Value of parameter**
Thickness	0.8mm
Blank size	960x600
Material yield	75%
Coefficient of friction	0.14%
Binder stroke	125mm
Binder holding force	60ton
Draw tonnage	145ton
Draw bead height	10mm
Die entry radius	6mm

A set of simulation run using auto form software were conducted out considering EDD grade material blank to determine effect of draw bead height, die shoulder radius and blank holder force. The simulation iterations were performed so as to investigate the thickness variation on drawn shell component. Three level and three factors L9 Orthogonal array is used to design the orthogonal array by using DOE and relevant ranges of parameters. Total nine iterations were conducted and nine thickness measurements were made on each drawn cup at different locations i.e., 9 point on the flange and then maintaining fix distance between adjacent points, as shown in Figure 5.

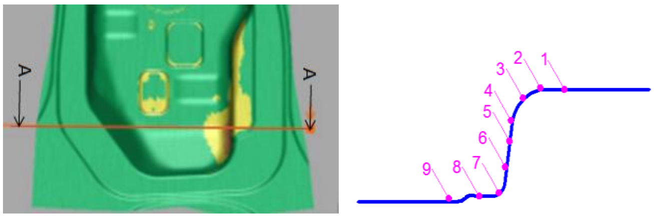
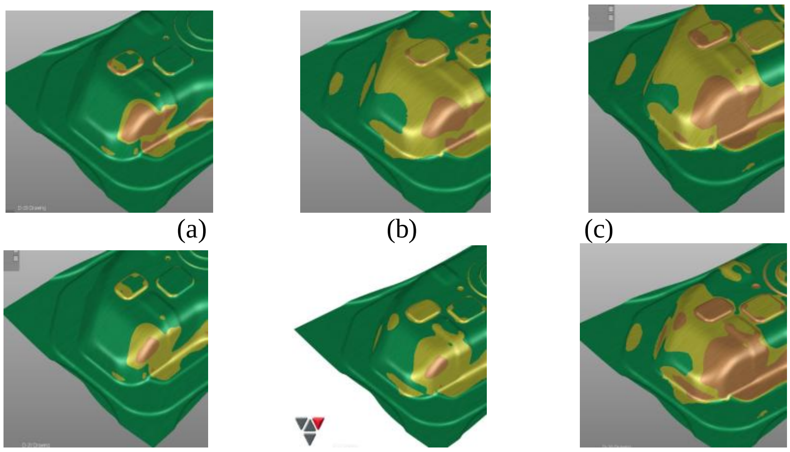


Fig 5 (a) Measuring positions after cutting the draw panel

(b) The split cup

The drawn shell component were cut into two halves as shown in Figure 5(a) and the thicknesses measurements were made at each point and the recorded thickness measurements were also represented graphically in Fig5(b) While evaluating the thickness, at each location three measurements were made and average values were computed. It had been observed from Figure 6(a) to Fig 6(i) that in the first run point 1 to last point 9 , the measured thickness value is following different trend. Fig 7 .



(a) (b) (c)

(d) (e) (f)

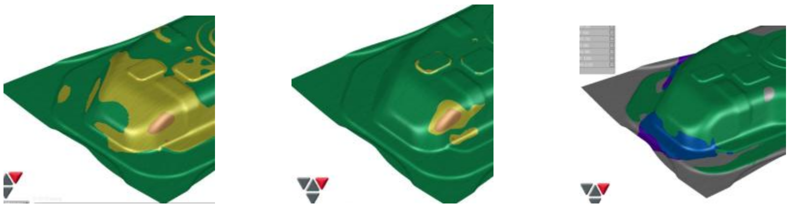

(g) (h) (i)

Fig 6 . Thinning plot iteration number 1 to 9

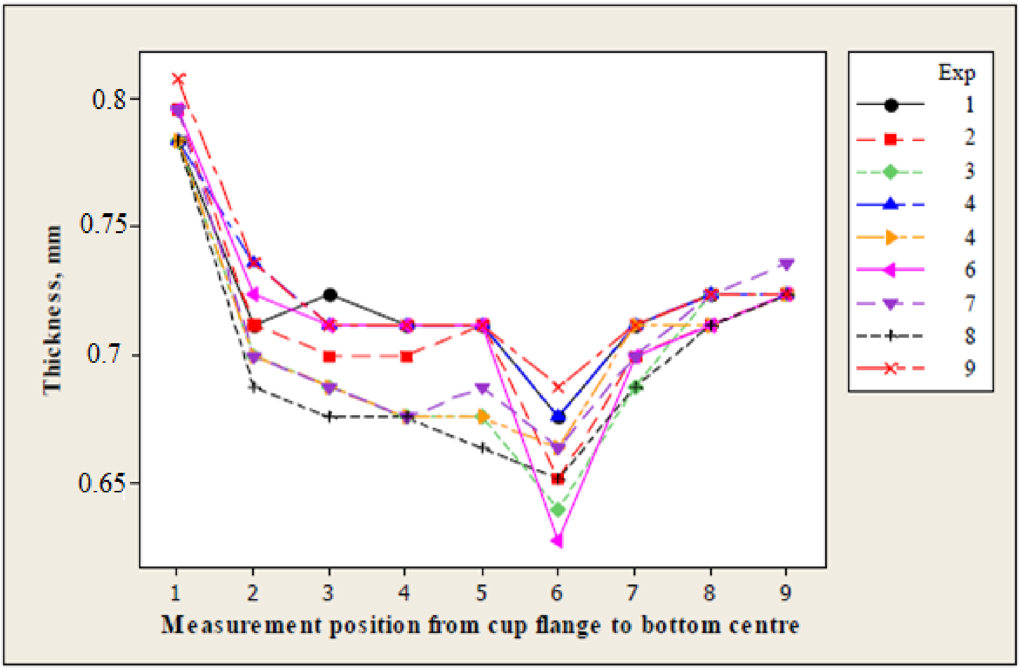


Fig. 7. Thickness Measurement [13]

Corresponding thickness values a measured at different locations as specified by points 1 to point 9 are givem in Table VI and Table VII .For Each value of Process parameter the thickness are found and ploted on graph shown in Fig 7 we have measured total 9 points. Using FEM analysis the corresponding thinning value is obtained. Which is converted in to thickness value.

TABLE VI: SIMULATION READING

Parameter Level				**Thickness value measured in different positions , mm**				
Exp Run	**Dbh mm**	**Rd mm**	**Fh KN**	**1**	**2**	**3**	**4**	**5**
1	5	5	55	0.73	0.66	0.67	0.66	0.66
2	5	8	60	0.75	0.66	0.65	0.65	0.66
3	5	10	65	0.73	0.65	0.64	0.63	0.63
4	8	5	60	0.73	0.69	0.66	0.66	0.66
5	8	8	65	0.73	0.65	0.64	0.63	0.63
6	8	10	55	0.75	0.67	0.66	0.66	0.66
7	10	5	65	0.75	0.65	0.64	0.63	0.64
8	10	8	55	0.73	0.64	0.63	0.63	0.61
9	10	10	60	0.76	0.69	0.66	0.66	0.66

TABLE VII: SIMULATION READING

Parameter Level							
Exp Run	**Dbh mm**	**Rd mm**	**Fh KN**	**6**	**7**	**8**	**9**
1	5	5	55	0.63	0.67	0.67	0.62
2	5	8	60	0.6	0.66	0.66	0.67
3	5	10	65	0.59	0.67	0.67	0.67
4	8	5	60	0.63	0.67	0.67	0.67
5	8	8	65	0.61	0.66	0.66	0.67
6	8	10	55	0.58	0.66	0.66	0.67
7	10	5	65	0.61	0.67	0.67	0.69
8	10	8	55	0.6	0.66	0.66	0.67
9	10	10	60	0.64	0.67	0.67	0.67

IV. RESULTS AND DISCURSION

A . Forming limit diagram with auto form after forming trial

As shows Fig 8 the various zones with various points falling in respective zones . The Red points indicate splits or Cracks. These are points located above the forming limit curve. These points are in the component region they must be eliminated in subsequent iterations so that the actual draw panel will be free from cracks and splits. Orange points indicate excess thinning on panel these points must be eliminated as the part produced will be defective. Green points represents safe zone free from all defects, Blue zone indicates compression, and Violet points indicate thickening wrinkling tendency. Forming limit diagram shown in Fig .8. The safety zone report is Coloured based on where a mesh element on the surface falls within a forming limit diagram. Therefore, red areas show where Wrinkles will occur, deep blue show cracks. The following figur e shows the safety zone plot of the given component. The result shows that there is no chance of tearing and the wrinkles can be controlled during try out.

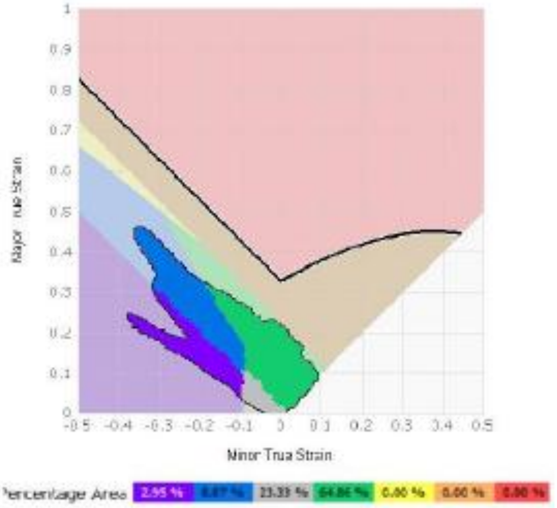


Fig 8 FLD plot showing major points of thinning.

B . Simulation result after forming trial

After performing 9 iterations we come to conclusion that we get optimum result. At 65 ton blank holding force. These results are shown in Fig 9 and Fig 10. The thickening or wrinkling tendency is reduced except at the corners this can easily be taken care off in die try out. As shown in Fig. 9. Simulation result for thinning is min -0.188 which is less than 20% and results are within limits so by minor try out correction we can correct the part.

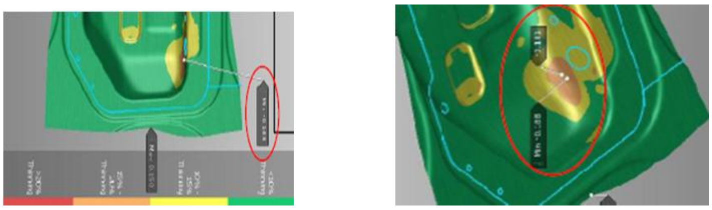
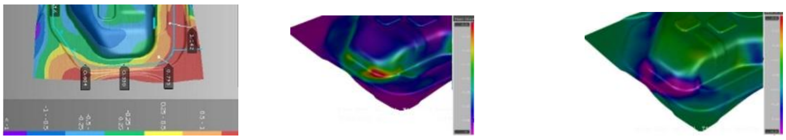


Fig 9 . Thinning plot Max thinning up to 18.8%



(a) (b) (c)

Fig 10 (a) spring back plot (b) major Strain (c) minor strain plot

As shown in Fig. 10 . Spring back observed is 1.42 mm max which is well inside the part boundary and 0.604max outside part boundary these values are acceptable values and by minor try out correction we can correct the part. Also major strain and minor strain are within permissible limit.

B . Experimental validation of simulation result

After studying various optimization method this Analysis of sheet metal forming it concludes that ,Optimized result are obtained by Autoform and Taguchi Analysis. DOE ,FMEA is used for finalization of setting optimized parameters. Various iteration are carried out to get optimized parameter .Those result are nearer but not fully optimized to get optimized parameters. So Taguchi Analysis is used. Optimizes result are obtained by Taguchi Analysis. And these results are verified by autoform analysis. Those verified result then used for Die Design. This results into Elimination of forming defects such as thinning wrinkle, scoring,. Also give good results in less time . Benefit of these Analysis is also that its saves times and actual trial cost. Final checking of component after Forming Process Using Following process parameters

- Blank holder force = 60T
- Blank size=970X625X0.8
- Type of press=500 Ton
- Shut Height=800 mm
- Blank holder stroke /Binder stroke =125 mm
- Master side=Die Master
- Friction Coefficient=0.14
- Draw bead height= 8mm
- Die radius =8mm

After setting this parameters and some shop floor trials ok component is manufactured in less time .The trial and error method of selection of process parameters can be replaced by FEM and DOE analysis

C . Die set manufactured for component trial

To carry actual trial on the press we designed and developed a draw panel die as shown in Fig.12 . Actual tryout was carried out at the specified 60 ton blank holding force all other parameters were kept as it is and we get successful result with ok panel after minor correction in die which reduced out total manufacturing time by 30% also number of actual trials were reduced to 3.

Fig 11 (a) Thinning measurements using point micrometer, (b)Component check on gauge

TABLE VV: ACTUAL VS SIMULATION READING

Simulation Result vs. Actual Result			
Point	Simulation Result	Actual Result	%Error
1	0.69	0.72	-3.75
2	0.72	0.7	2.5
3	0.65	0.67	-2.5
4	0.72	0.69	3.75
5	0.54	0.58	-5
6	0.55	0.63	-10
7	0.72	0.7	2.5
8	0.67	0.66	1.25
9	0.78	0.76	2.5

Also the part was tested on a checking fixture as shown in Fig.11. For its dimensional accuracy the overall geometry of part was check on cmm machine through cad comparison and we get satisfactory results .The final ok panel is shown in Fig.11.

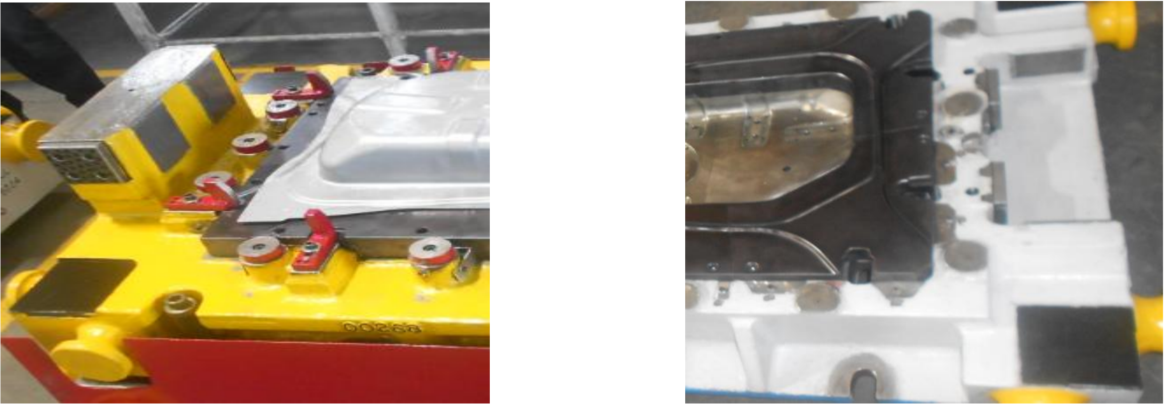


Fig 12 (a) Lower die shoe (b) Upper die shoe

V. CONCLUSION

The main defects in draw panel cracking and wrinkling can be predicted well before by simulation software .error measured between simulation software and actual reading is less than 10%.. After evaluating the sheet metal forming defects using FMEA it is found that the process parameter Blank holder force , die entry radius ,bead height have highest RPN value 288. Hence this process parameters are critical and need to be optimized. By using FEM analysis Autoform and performing DOE Taguchi the optimum value of process parameter are found as draw bead height of 8 mm, die shoulder radius of 8 mm and the blank holding force of 60 kN. This results are validated by actual shop floor trials. Using optimized FMEA ,DOE & simulation methods , implementing those results the defect in metal forming can be predicted well advance and eliminated .

ACKNOWLEDGMENT

The author would like to thanks Prof.V.S.Aher, PG Coordinator Amrutvahini COE for his valuable support and guidance.

REFERENCES

[1]. S.Sulaiman,M.K.Ariffin and S.Y.Lai, “Spring Back Behaviour In Sheet Metal Forming Fpr Automotive Door”, ELSEVIER AASRI Conference On Modelling Identification and Control , 224-229 , 2012.
[2]. Olaf Schenk and M. Hillmann ,“Optimal design of metal forming die surfaces with evolution strategies”, Elsevier science direct Computers and Structures , 1695–1705, 2004.
[3]. Jae-Jun Lee and Gyung-Jin ,"Optimization of the structural and process parameters in the sheet metal forming process",Springer,Struct Multidisc Optim ,vol35,571–586, 2008.
[4]. Claes Arwidson, “Numerical simulation of sheet metal forming for high strength steel,” Licentiate thesis, Lulea university of technology, 97187 Lulea, Sweden, Aug. 2005.
[5]. G.Pantazopoulos and G.Tsinopoulos, “Process Failure Modes and Effects Analysis (PFMEA),” ASM International, JFAPBC, vol. 5, pp. 1547-7029, April. 2005.
[6]. R.S.Mhetre and R.J.Dhake,“ Using Failure Mode Effect Analysis in a Precision Sheet Metal Parts Manufacturing Company,” Int. Journal of Applied Sciences and Engineering Research, Vol. 1, No. 2, 2012.

[7]. J.Byeongdon , K.Sangyun, K.Suhee and M.YoungHoon, “FMEA for the reliability of hydroformed flanged part for automotive application,” Springer Journal of Mechanical Science and Technology, vol 27 , 63-67,2013.
[8]. K. Debray, Y.Li & Y.Guo, “Parametric design and optimization of addendum surfaces for sheet metal forming process”, Springer, International Journal Mater Form 6 ,315–325, 2013.
[9]. H. Zein, M. Sherbiny, M.Rabou, M. Shazly," Effect of Die Design Parameters on Thinning of Sheet Metal in the Deep Drawing Process" ,American Journal of Mechanical Engineering, Vol. 1, 20-29,2013.
[10]. M.Bonte , A.Boogaard and J.Huetink ,“An optimisation strategy for industrial metal forming processes”, Springer Struct Multidisc Optim ,35:571–586, 2008.
[11]. Yaning Li , M.Luoa , J.Gerlach and T.Wierzbicki , “Prediction of shear-induced fracture in sheet metal forming” , ELSEVIER , Journal of Materials Processing Technology , vol 210,1858-1869,2010.
[12]. Duan Lei, Miao Dehua, Cai Yujun, Qu Zhoude and Li Zhen, “Research on Numerical Simulation for Automotive Panel Forming and Spring-back Based on Dynaform” ,IEEE Third International Conference on Measuring Technology and Mechatronics Automation,55-58,2011.
[13]. Reddy A , Rajesham . S, Reddy .P ,Kumar.T , Goverdhan.J, “ An experimental study of process parameters in deep drawing using Taguchi technique” , International Journal Of Engineering, Science and Technology , Vol.7 No1, 2015.