International Journal of Latest Engineering Research and Applications (IJLERA) ISSN: 2455-7137

Volume – 02, Issue – 06, June – 2017, PP – 86-92

APPLICATION OF FEA IN B-PILLAR REINFORCEMENT FORMING SIMULATION

Karale Nilesh S[1], Gandigude Ashish[2]
[1]*(Zeal college of Engineering and Research, Pune University, India)*
[2]*(Professor, Zeal college of Engineering and Research, Pune University, India)*

Abstract: The development of the automotive body represents a major challenge for all manufacturers as they continuously work to reduce the time and cost of bringing a new vehicle to market. We are trying to use computer simulation for manufacturing processes in order to reduce physical iterations. Stamping process of formed components involves consideration of various parameters such as material properties, elimination of defects such as wrinkling, tearing, spring-back effect etc. Simulation of stamping process involves accurate prediction of highly nonlinear deformations and contact analysis. During forming process, the material & process parameters have a significant effect on the quality of the final part. This study aims at developing a CAE-based process design for sheet metal forming of an automotive component in order to produce a good quality formed product. The objective is to investigate utilization of capabilities of Abaqus CAE for the tool design of an automotive member with high strength steels. The base tool model of the cup shape part is considered for the validation of material properties. Stamping of this automotive component has been simulated in Abaqus using contact analysis parameters and case study of B-pillar reinforcement pillar carried out.

Keywords: Blank holder force ,Drawing, FEM, Sheet metal simulation, Thinning.

1. INTRODUCTION

Deep drawing process is capable of forming circular shapes such as circular cups or even rectangular shapes or shell-like containers. Deep drawing products in modern industries usually have a complicated shape, so these have to undergo several successive operations to obtain a final desired shape.

While designing press tool, the tool designer's goals are

- To reduce the tool cost
- Estimate the blank shape and size
- Decide the number of stages
- Calculate the required press tonnage
- To maintain the specified thickness of the formed sheet
- To achieve the required finish - wrinkles and tears must be avoided

The tool designer has to choose a combination of process parameters that will yield a component that matches these specifications. And this is where simulation helps and shows its power to reduce tool cost .Stamping includes a variety of sheet-metal forming manufacturing processes, such as punching using a machine press or stamping press, blanking, embossing, bending, flanging, and coining. Defects that occur during deep drawing of sheet metal can be controlled by careful regulation of process factors. Excessive thinning in areas of the sheet metal is also an unwanted defect. Causes of these are mostly too high or improper force distribution and material considerations. FEA can be used as effective tool which will be help tool designer to create accurate tools which will lead to reduction on physical tool tryouts.

2. LITERATURE REVIEW

Chandra Pal Singh, et. al had done review on deep drawing process parameters [1]. Krupal Shah, et. al had done the study to determine the most critical process parameters that cause defects and thinning in the blanks. [2].Wifi, et. al had studied some aspects of blank-holder force schemes in deep drawing process [3].Jing Han,et. al had investigated optimum drawing process by changing circumferentially the blank holder forces, to draw earless circular cups from anisotropic blank sheet [4]. Y. Marumo, et. al had presented a Finite Element model developed for the 3-D numerical simulation of sheet metal deep drawing process by using ABAQUS/EXPLICIT Finite Element Analysis program [5]. Gyadari Ramesh, et. al aimed to study and perform analysis of optimization of blank holding force developed for cup drawing operation by using explicit finite element package LS DYNA [6]. Ketaki N. Joshi, et. al presents an investigation of the effect of die draw radius, sheet thickness and blank holder force on the variation in wall thickness of a deep drawn cup using finite element simulations [7]. H. Zein, et. al studied on thinning and spring back prediction of sheet metal in the deep drawing process [8]. R. Venkat Reddy, et. al studied on effect of die corner radius and punch corner radius on wrinkling and fracture limits in deep drawing of cylindrical cup [9]. Dr.R.Uday Kumar[10] have done study on study on deep drawing and spinning process in sheet metal forming. F. Ayari, T. Lazghab, E. Bayraktar [11] presented a paper which deals with the FEA of the sheet metal forming process that involves various nonlinearities.

3. FEA TOOL: ABAQUS EXPLICIT

Finite Element Analysis (FEA) simulates a physical part or assembly's behaviour by dividing the geometry of the part into a number of elements of standard shapes, applying loads and constraints, then calculating variables of interest – deflections, stresses, temperatures, pressures, etc. ABAQUS is a suite of finite element analysis modules. The heart of ABAQUS are the analysis modules, ABAQUS/Standard and ABAQUS/Explicit, which are complementary and integrated analysis tools. Summary of steps involved in FEA is as follows.

- Import the CAD model of the final component
- Fill holes if they are to be created after the forming
- Generate a shell mesh of the component
- Orient the component so the draw direction is along the z-axis
- Check for undercuts and tip the component if necessary
- Specify the material properties – indicative values are adequate
- Specify the thickness of the component
- Identify areas that are to be trimmed, if any, so that the solver can take these into account when estimating the blank size
- Specify analytical drawbeads, if any
- Run the analysis
- Inspect the results – thinning, formability and press tonnage.
- Thinning percentage is best viewed as a contour, while formability is best viewed as the FLD.

Abaqus provides few control parameters which help to correlate the FEA results with actual test like stabilization factor, use of damping factor, adaptive automotive stabilization.

4. EXPERIMENTAL VALIDATION FOR MATERIAL

Circle grid analysis carried out to understand the material behaviour for given loading conditions. The experimental model used to determine the forming limit diagram is simulated in Abaqus in order to validate the results of abaqus. A circular sheet of metal was formed in abaqus. The material properties for the simulation were as discussed above.

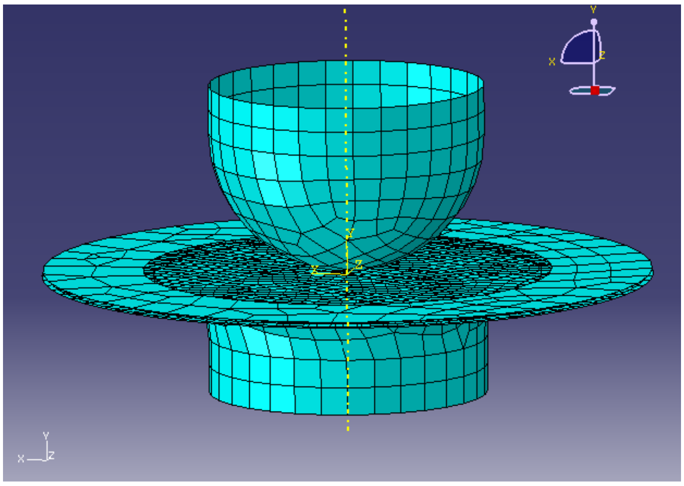

Fig. 4.1. Simulation of simple component in Abaqus

4.1 COMPARISON OF THE SIMULATION AND THE EXPERIMENTAL RESULTS

Figure 4.3 shows the failure location comparison between experiments and simulations. It is clear from the figure that failure locations predicted using FE simulations (based on thickness gradient criterion) are in good agreement with experimental observations.

Fig. 4.2. Experimental Setup for FLD determination

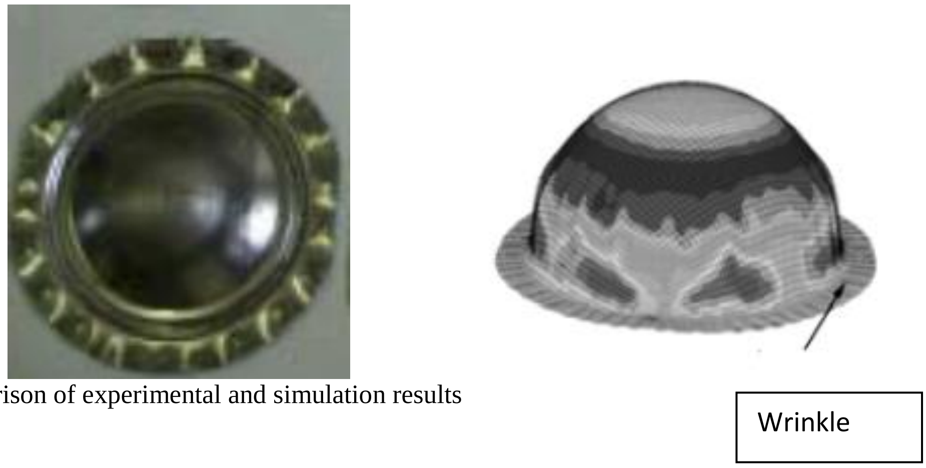

Fig. 4.3. Comparison of experimental and simulation results

5. B-PILLAR REINFORCEMENT OF CAR

The B-pillar of a car is the structure located between the front and rear doors of the cab. It does not only house electrical wiring and connection spots for the passenger seat belts, but it provides structural support for the cab in case of a side collision or rollover of the vehicle. B–pillar is a structure that is seen on the two sides of the car, which is held by the roof rail and the bottom rail of a vehicle frame. It acts as a rigid support for the body frame to resist the forces acting on the vehicle due to side impact and in case of roll-over of the vehicle. The B-pillar is provided with reinforcement to reduce the deformation due to side impact force.

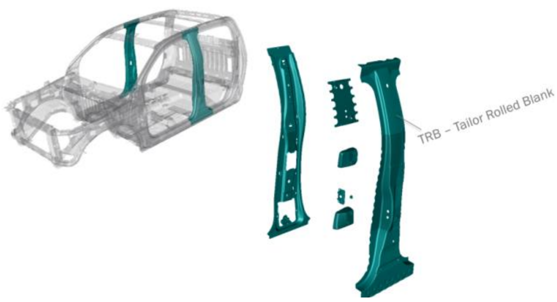

Fig. 5.1. B-Pillar reinforcement

5.1. PURPOSE OF SIMULATION

For new design of automobile B-pillar, we analysed the b pillar reinforcement for reduced thickness of 1 mm using ABAQUS software. The purpose of simulation is,

1. To finalise the blank size
2. To minimize wrinkle tendency in formed final part
3. To predict thinning in final part and intermediate stage.
4. To check the springback after forming operation
5. To evaluate straining in formed part.

5.2. PROCESS PLAN FOR FINAL SHAPE

For manufacturing of B-pillar reinforcement multi stage forming is used. Following sequential operations are performed on sheet metal to get the final component shape.

OP 05 :- Blank and Pierce
OP 10 :- Draw
OP 20 :- Trim and Pierce
OP 30 :- Flange and Restrike
OP 40 :- Trim, Pierce & Flange
OP 50 :- Cam end trim & Pierce

5.3 FINITE ELEMENT MESH

Figure 5.2 Schematic exploded view of FE model used to during modelling. All tooling is modelled as discrete rigid, meaning that tool deformation is prevented during the analysis. Reference points are defined to the tooling geometry in order to give the boundary conditions.

Typically, the blanks are meshed with element size 2.5 mm same size as used for circular grid during experiment. It capture the complex curvature of the tooling as accurately as possible with only a minimal adverse effect on computation time.

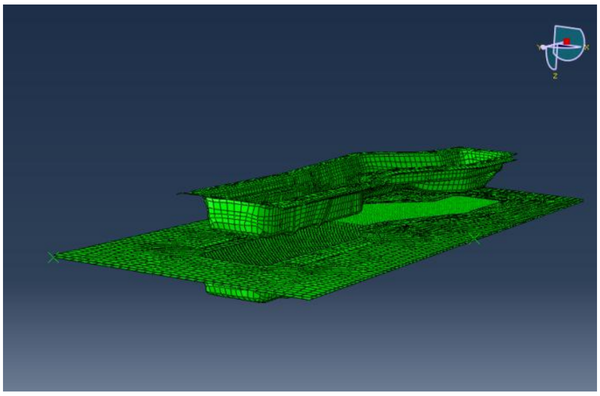

Fig. 5.2. Schematic of the FE setup

5.4 BOUNDARY CONDITIONS AND TOOLING MOTION

There are no nodal single point constraint (symmetry) boundary conditions imposed on the deformable HSS. Their deformation and motion is solely defined by contact with the surrounding parts. The tools themselves are constrained to minimize the motion of the tools in the press. The Die is prevented from moving, while the Punch and Blank holder are constrained to move solely in the Z axis. A prescribed tool motion is defined for the Punch. During forming, the die cavity is lowered over the blank which rests on the binder plate. The sandwich blank is then stretched over the fixed punch as the die tooling continue its motion in the negative z-direction well beyond the necking. The blank holding or clamping force was prescribed as an applied load in the negative Y-direction. All the parts are positioned in contact at the start of the simulation to eliminate any abrupt acceleration. The amplitude applied to the displacement of the Punch is according to the cycloidal law in order to attain convergence.

- Punch: Punch is kept free in Z direction and kept fixed in all five direction.
- Die clamped in all directions.
- Blank is kept free in all directions.
- Blank Holder : Blank holder allowed to kept free in Z direction and fixed in all five direction.

Blank size used for this is 1400 mm * 678 mm. Blank weight is 7.80 Kg and actual part weight is 1.81 Kg having yield of 24.3%.

5.5 RESULTS AND DISCUSSION

The simulation of the deep drawing process with the above defined parameters gives a perfectly formed blank as shown in fig the contour plot of von-mises stresses sows uniform distributed stresses. A complete deformed stress pattern is observed. A perfect shape is seen to be formed after the completion of explicit dynamic step.

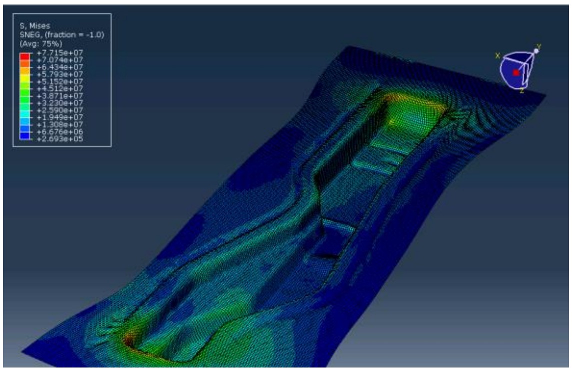

Fig. 5.3 Fully deformed component after simulation

From the contour plot of Vonmises Stresses the area of maximum stresses can be identified. It is located at the corner radius. The element with maximum stress is identified and the corresponding element no is noted. The strain path of the respective element is plotted ant it seen that the strain path is below the Forming limit curve. Thus it can be concluded the component is well within the permissible limits.

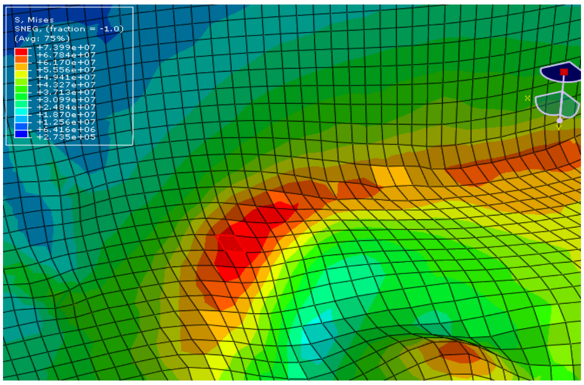

Fig. 5.4 Element number: 33527

After simulation of the blank it can be seen that there is a region where the stress level is very high. One of the element of this stress region is identified and the major and minor strain of this element are plotted. The plot represents the actual condition of the element. If the strain plot is below the FLC curve then the region is within permissible limits or it gets tear.

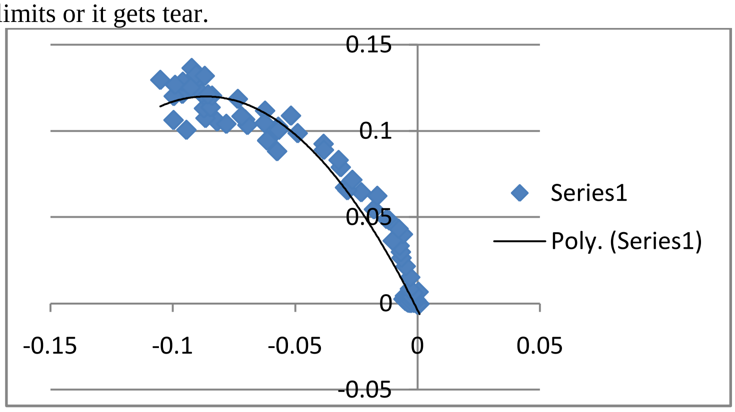

Fig. 5.5 Forming Limit Curve

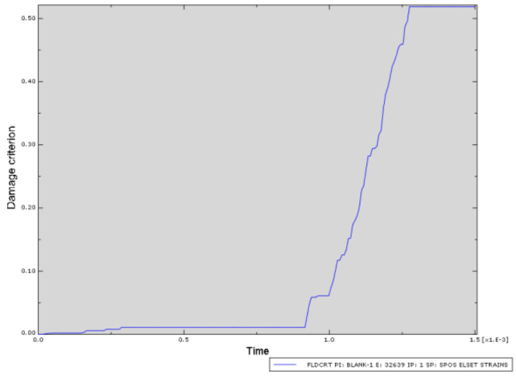

Fig. 5.6 Damage initiation criterion of element no:33639

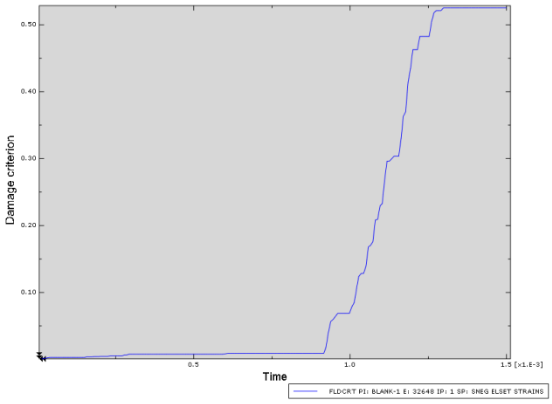

Fig. 5.7 Damage initiation criterion of element no:33639

The Damage initiation criterion of critical element 33693 shows its value is 0.5. which is wll below the permissible value i.e. 1. Thus from the damage initiation criterion it can be said that the component is safe.

5.6 Optimization of the Component:

Optimization of the component can be done by changing the parameters as below. Punch displacement amplitude can be varied to obtain different results. Also the Blank holding force and the coefficient of Friction values can be changed to obtain optimum conditions for stamping process.

Punch Displacement
Amp1
Amp2
Amp3

Blank Holding Force
1.00E+07
1.00E+06
1.00E+05

Table 5.1 Variation in the parameters

The above variations can be one in the parameters as shown in table 5.1. By changing the parameters in random an optimum parameters can be found out which give a perfect solution of the stamping conditions.

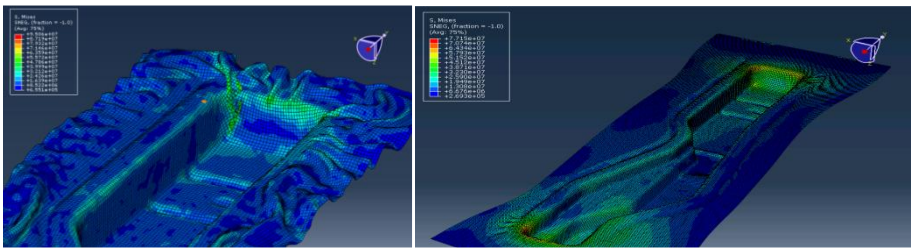

Fig. 5.8 Comparison of the wrinkled component with that of the clear one by changing Blank Holding Pressure.

6. CONCLUSION

Metal forming process in which a sheet metal blank is radially drawn into a forming die by the mechanical action of a punch. Deep drawing is used in various industries. Deep drawing comes under sheet metal forming process because it uses a metal sheet as a blank to deform it to another or desired shape. During this process various defects may occur like wrinkling, fracture, thinning and spring back. Defects that occur during forming of sheet metal can be controlled by careful regulation of process factors like blank holder force, radius on punch and coefficient of friction needs to be controlled.

Based on the analysis performed on B-Pillar Reinforcement of car and then the optimization of the component. Some conclusions can be drawn. Following are the conclusions.

1. Forming Limit Diagram Damage initiation Criterion. The Forming Limit diagram criteria of highly stressed element are 0.5 which is below 1. In accordance with the FLD criteria theory it should be less than 1 to be safe. Thus the component is safe.
2. On plotting of Strain Paths comparison of FLC of the highly stressed element and the FLC diagram of same clearly indicates that the highest major true strain and minor true strain coordinates of the element are well below the corresponding coordinated of material.
3. It is been observed while optimization that if the coefficient of friction value is below 0.1 the small amount of wrinkle may occur at the corner radius and if the Blank Holding Pressure is below 1e6 the total component gets wrinkled together.
4. It is been observed that the simulation results of simple circular sheet and that of actual stamping product are equal thus we can say that the simulation results are correct. This in turn reduces the major defect form the component and increase in safety zone. Thus, the defined objectives of the research have satisfied.

REFERENCES

[1] Chandra Pal Singh, Geeta Agnihotri (2015), "Study of Deep Drawing Process Parameters: A Review", *International Journal of Scientific and Research Publications, Volume 5, Issue 2, ISSN 2250-3153*.

[2] Krupal Shah, Darshan Bhatt, Twinkle Panchal, DhruvPanchal, Bharat Dogra, "Influence of the Process Parameters in Deep Drawing", *International Journal of Emerging Research in Management &Technology,Volume-3, Issue-11, ISSN: 2278-9359*.

[3] Wifi, A. Mosallam (2007), "Some Aspects of Blank-holder Force Schemes in Deep Drawing Process", *Journal of Achievements in Materials and Manufacturing Engineering, Volume 24, Issue 1,pp. 315-323*.

[4] Jing Han, Koetsu Yamazaki, Shinya Makino and Taketo Shirasawa (2013), "Optimization of Deep Drawing Process for Circular Cup Forming", *10th World Congress on Structural and Multidisciplinary Optimization , Orlando, Florida, USA, pp. 1-6.*

[5] Y. Marumo, H. Saiki, L Ruan, "Effect of Sheet Thickness on Deep Drawing of Metal Foils", *Journal of Achievements in Materials and Manufacturing Engineering, Volume 20 Issues 1-2 January-February 2007, Pp. 479-482.*

[6] Gyadari Ramesh, Dr.G.Chandra Mohan Reddy (2013), "Analysis of Optimization of Blank Holding Force In Deep Drawing By Using LS DYNA", *International Journal Of Engineering Research and Applications (IJERA), Volume 3, Issue 4, ISSN: 2248-9622,pp.1975-1995.*

[7] Mrs. Ketaki N. Joshi, Dr. Bhushan T. Patil, Mr. Sunil Satao, Dr. Chandrababu. D (2014), "Optimization of Variation in Wall Thickness of a Deep Drawn Cup using Virtual Design of Experiments", *IRACST – Engineering Science and Technology: An International Journal (ESTIJ), Volume 4, Issue 5,ISSN: 2250-3498.*

[8] H. Zein, M. El Sherbiny, M. AbdRabou, M. El Shazly (2014), "Thinning and Spring Back Prediction of Sheet Metal in the Deep Drawing Process", *Materials and Design 53 ,pp. 797–808*.

[9] R. Venkat Reddy, Dr. T. A. Janardhan Reddy, Dr. G. C. M. Reddy (2013), "Effect of Die Corner Radius and Punch Corner Radius on Wrinkling and Fracture Limits in Deep Drawing of Cylindrical Cup", *International Journal Of Emerging Technology and Advanced Engineering, Volume 3, Issue 3, March 2013.*

[10] Dr.R.Uday Kumar, "A Study On Deep Drawing And Spinning Process In Sheet Metal Forming", *IJAIEM, Volume 2, Issue 2, February 2013, ISSN 2319 – 4847, Pp. 170-175.*

[11] Najmeddin Arab, AbotalebJavadimanesh, "Theoretical and Experimental Analysis of Deep Drawing Cylindrical Cup", *Journal of Minerals and Materials Characterization and Engineering, 2013, 1, 336-342.*

