International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-04, Issue-04, July 2018

Finite Element Analysis of Automobile Roof Header Manufactured by Stamping Process

[1]Kete Krishnaraj Shivaji, M.E. Student, Department of Mechanical Engineering,
BMIT, Solapur, kkete@rediffmail.com
[2]Prof. Mithun D. Shah, [3]Hrushikesh B. Kulkarni, [4]Prof. Shailesh A. Nehatrao,
[2,3,4]Assistant Professor, Department of Mechanical Engineering, NBNSCOE, Solapur, Maharashtra.

Abstract This work presents a methodology based on numerical simulation to predict contact pressure distribution, using Ansys a simulation software, and investigates the influences of various processing parameters in stamping process used in manufacturing of automotive roof header. The results are captured for von mises, displacements, plastic strain and contact pressure. The contact pressure picture shows variation of contact pressure along the geometry. This contact pressure prediction helps in proper design of stamping tools to reduce errors in the stamping process. Also plastic pictures help in predicting the region of crack formation and higher residual stress formation which are the sources for reduction of life of the component. From work it is observed that the total stress found 995 Mpa is more than the yield strength of the material, hence the sheet deforms into the shape of the die cavity, The elastic and plastic strain observed in the analysis is within limit i.e. (0.65 & 0.55) which is less than 1 mm/mm hence the analysis is correct.

***Keywords* — Finite Element Analysis (FEA), Automobile roof header, Stamping Process.**

I. INTRODUCTION

Stamping (also known as pressing) is the process of placing flat sheet metal in either blank or coil form into a stamping press where a tool and die surface forms the metal into a net shape. 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 [1]. This could be a single stage operation where every stroke of the press produces the desired form on the sheet metal part, or could occur through a series of stages. The process is usually carried out on sheet metal, but can also be used on other materials, such as polystyrene. Progressive dies are commonly feed from a coil of steel, coil reel for unwinding of coil to a straightener to level the coil and then into a feeder which advances the material into the press and die at a predetermined feed length. Depending on part complexity, the number of stations in the die can be determined.

The process of stamping is traditionally associated with the auto industry, and all of its requirements for specially shaped panels and parts. However, as technology advances, stamping and pressing are increasingly being used to produce medical devices and electronic products as well. The Precision Metal forming Association recently tracked a sharp business increase for the industry, even during this most recent economic downturn.

As for the actual processes, in stamping and pressing machines, sudden impact and high forces are par for the course — which makes adequate damping essential.

Pressing and molding parts from steel also benefit from automation, particularly where raw rolls and sheets weigh many thousands of pounds.

In stamping operations, sheet metal is formed into a desired shape by pressing it in a hydraulic or mechanical press between suitably shaped dies. As a predominant manufacturing process, sheet metal forming has been widely used for the production of automobiles, aircraft, home appliances, beverage cans and many other industrial and commercial products.

A major effort till date on stamping processes monitoring has been focused on investigating variations in the press force. Given that the press force itself is an integral of the contact pressure distribution over the die and binder contact interfaces, it is conceivable that defects may be better identified by analyzing the contact pressure distribution directly at the tooling work piece interface.

Until a few years ago design of metal forming tools was mostly based on knowledge gained through experience and designing the optimum tool often required a protracted and expensive trial and error process. Today even in the early design phases simulation of sheet metal parts forming processes are performed using Finite element analysis [2].

 DOI : 10.18231/2454-9150.2018.0559

Roof header made of Al-6061 alloys an important component in a passenger car. It is the roof headers profile that plays an important role in giving the befitting elegance to an automotive vehicle. The key features of the roof header are built in using the stamping operations of a sheet metal of light alloys like Al. 6000 series. During the stamping process it's of utmost importance to ensure all the key features in the roof header die gets impressed to the closest tolerance. It is equally important to ensure good surface finish during the stamping operation.

All the above factors discussed in this section are mainly controlled by the stamping process parameters. Ram velocity, clamp holding pressure, the uniformity in the thickness of the sheet metal, the design of the die and the punches are some of the important parameters to control in order to ensure high quality roof header [3].

In the present work, two types of analysis are considered. One three dimensional symmetric object is analyzed for sheet metal formation for stamping process and a two dimensional analysis for irregular shaped objects. Initially the geometries for two dimensions and three dimensions are built and later meshed for finite element calculations.

In three dimensional analyses, due to symmetry, quarter geometry is considered due to computational complexity to reduce the solution time. Also temperature effect is considered in the problem to find the stamping operational load. The problem is converged at different steps and results are captured for stress and plastic strain effects. The result shows slight variation of numerical and finite element results.

Further two dimensional analyses for stamping also shows higher number of steps for stamping operation. The results are captured for von-mises, displacements, plastic strain and contact pressure. The contact pressure picture shows variation of contact pressure along the geometry.

This contact pressure prediction helps in proper design of stamping tools to reduce errors in the stamping process. Also plastic pictures help in predicting the region of crack formation and higher residual stress formation which are the sources for reduction of life of the component.

II. LITERATURE REVIEW

This contact pressure prediction helps in proper design of stamping tools to reduce errors in the stamping process. Also plastic pictures help in predicting the region of crack formation and higher residual stress formation which are the sources for reduction of life of the component.

DynaForm software is used to simulate the process of punching the automobile beam, which provides a basis for the design of the actual automobile crossbar stamping die. The design of the actual stamping molding parameters provides theoretical guidance.

A finite element analysis model was established and analyzed by DynaForm software. Through the analysis of forming limit diagram, material flow chart and thinning rate graph, it can be seen that the radius, blank holder force, friction coefficient, punching speed and drawing resistance of convex and concave die have great influence on the forming process. In the actual production, should be based on simulation, select a reasonable value [4].

Tube hydroforming (THF) is currently an active area of development in the automotive industry on account of its advantages it offers in comparison to other manufacturing processes. In this paper, four simulations were carried out having the same cross sectional area but different axial feeding and load paths to improve the process design in THF process, in order to help establish the feasibility of the THF processes for specific components and have the potential to reduce the number of forming trials required to standardize the process for production of high quality hydroforming components.

In addition, the studies also focus on development of an experiment simulation procedures for the entire hydroforming process. The simulation results focus on the % reduction thickness. It has also focus on the axial feed and load paths which becomes an important and useful method for planning the process design analysis, design of forming process and also planning of the tool. Since they play an important role and also determine accordance of forming limits such as wrinkling, bursting and crushing .The geometry model is analyzed base on different axial load and load paths. The key issues include geometry modeling, materials selection, meshing, boundary conditions, load definition and contact between the tube and the dies. On the base of the simulation, an optimized process parameter combination has obtained and has been verified by the instrument panel frame hydroforming experiment [5].

The objective of this study is to explore and identify vehicle body design countermeasures to meet the proposed upgrade of the FMVSS 216 standard. Finite Element (FE) methods in LS-DYNA solver are used as a tool to evaluate the performance of each countermeasure. A generic FE model of a sport utility vehicle developed at the National Crash Analysis Center (NCAC/GWU) is used as an exemplar vehicle.

Traditional countermeasures such as material grade and gauge change are applied to this exemplar vehicle to study the benefit of added mass in increasing the roof crush resistance. In addition, innovative countermeasures in the form of composite structural reinforcements are considered in this study. Component level three point bend tests are conducted with and without the composite reinforcements to evaluate the benefit of composite body solutions for roof crush applications. This paper provides an overview of the

 DOI : 10.18231/2454-9150.2018.0559

substantial benefits of utilizing Composite Body Solutions® (polymeric structural foam materials and their designs) in significantly improving roof crush performance [6].

Vehicle design has been driven by sales and marketing factors for many years, with a few exceptions like the brutish looking Hummer Sport Utility Vehicle (SUV) which was marketed for its structural safety. While it's partially true for public customers, consumers of work trucks commercial operations like in the oil, gas and mining industry are a large fleet consumer for dual cab transportations and an even larger segment for simple single cab work trucks.

These vehicles are designed to meet the minimum specifications for safety equipment and structural tolerances and therefore can be inadequate in various crash modes. Most manufacturers focus on sales in the US where regulations are design determinants, but outside the US, there are some times little or no minimum safety design regulations. The Mitsubishi Triton and the Toyota Hilux 4 and 2 door pickup vehicles, all of which are not sold in the US, require structural reinforcement for safe operation in these non-ordinary environments. This paper focuses on the design and performance of these work trucks and the means by which rollover safety can be measured and significantly improved [7].

The objective of this study was to analyze the influence of different platen angles of the Federal Motor Vehicle Safety Standard (FMVSS) 216 test and the influence of the resulting load application on roof deformation patterns. Of particular interest was how the strength of the connection between the A- pillars (roof header) influenced the roof crush performance. A Finite Element based study using the Ford Explorer altered the roof header design and applied High Strength Steel (HSS).

Pitch and roll angles of the loading device were varied. When the pitch angle increases, the strength- to- weight ratio (SWR) decreases for all roll angles. The worst case scenario was with a platen angle of 25° roll and 10° pitch. Once the connection between the A- pillars was improved, the SWR increased for high roll angles. The combination of a redesigned roof header and the use of HSS led to the highest SWR improvements ranging from 12.9% to 23.1%. The fixed platen angles for FMVSS 216 test appear to provide little incentive to improve the overall roof structure by load transfer to the opposite side A- pillar. Actual rollover accident subjects the roof to a variety of loading angles more extreme than the FMVSS 216 test condition [8].

Most rollover safety research considers the vehicle in an inverted (upside-down) position as the major cause of serious injuries. For example, neck injuries mostly occur when the vehicle roof strikes the ground.

However, recent studies have identified that significant injuries can occur to restrained and contained occupants during other phases of the rollover event, including when the vehicle is on its side and upright. Some injurious impacts between the occupant and vehicle interior such as torso injuries are likely related to changes in their relative velocities (Δv).

Such velocity changes are believed to be associated with a higher number of quarter turns during a rollover event. This study seeks to identify the role of roof shape for Sports Utility Vehicles (SUVs) in the potential for exacerbating vehicle-occupant Δv and thus the potential risk for associated occupant injuries. Detailed computer finite element model (FEM) rollover crash analysis, determining the kinetic energy, translational velocity and roll rate changes of a simplified proxy SUV FEM vehicle that occur during a two roll event for two vehicle roof shapes (square and rounded), is presented.

The vehicle's kinetic energy, translational velocity and roll rate for each considered roof shape during the rollover are compared. It is hypothesised that the rounder roof design results in a less severe change for each of these variables particularly towards the end of the rollover event, thereby reducing occupant-vehicle impact severity. This work is part of an ongoing study needing further work to confirm these findings in real-world SUV rollovers [9].

III. SCOPE AND OBJECTIVE OF WORK

It has been observed that very few studies have been made on application of simulation software in analysis of automobile roof header which is manufactured by stamping operation [10-13]. Especially the effect of pressure distribution at different temperature and effect of major process parameters such as critical regions of possible cracks, load requirements, deformation, binder pressure.

This work presents a methodology based on numerical simulation to predict contact pressure distribution, using simulation software, and investigates the influences of various processing parameters [14,15].

The results are captured for von mises, displacements, plastic strain and contact pressure. The contact pressure picture shows variation of contact pressure along the geometry. This contact pressure prediction helps in proper design of stamping tools to reduce errors in the stamping process. Also plastic pictures help in predicting the region of crack formation and higher residual stress formation which are the sources for reduction of life of the component [16].

1. Estimation of thermal effects on structural deformation and resultant stresses
2. Identification of critical regions of possible cracks
3. Identification of load requirements

 DOI : 10.18231/2454-9150.2018.0559

4. Effect of Deformation on stress generation.
5. Generation of contact pressure development in the stamping process
6. Possible problems with two dimensional and three dimensional process

Due to the advances in finite element based numerical software, work is required to identify the finite element application in these problems. Finite element simulation helps in avoiding the prototype built and costly setups. Also it reduces the solution time along with internal details which are not possible with practical built up models.

IV. METHODOLOGY

Here two ways of stamping process are simulated. In the first process a three dimensional approach is considered for the contact pressure simulation. Later a two dimensional analysis is considered for the contact pressure development in the stamping process.

1. Initial built up of geometry of movable die, fixed die and sheet metal
2. Meshing with three dimensional elements
3. Contact pair creation between fixed die, movable die and sheet metal
4. Study of Material Properties.
5. Applying Boundary Conditions such as Punch velocity, holder force.
6. Solving the problem with different temperature dependent material data.
7. Analyzing the problem
8. Results presentation

V. FINITE ELEMENTS ANALYSIS

The stamping process is metal deformation process under the load. To deform the material the load applied should be greater than plastic strength of the material i.e. yield strength.

1. **Model:** The model is created in Design Modeler which is the application of Ansys. The model consists of a die, a punch and a sheet which is to be deformed.

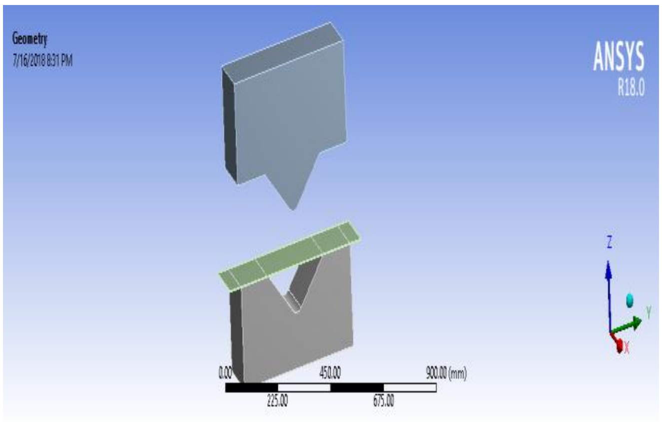


Fig1. Model in Design Modeler

The sheet is held on the die, and punch is displaced in the die cavity with some velocity, so that the sheet deforms with the kinetic energy of the punch.

2. **Boundary conditions:**

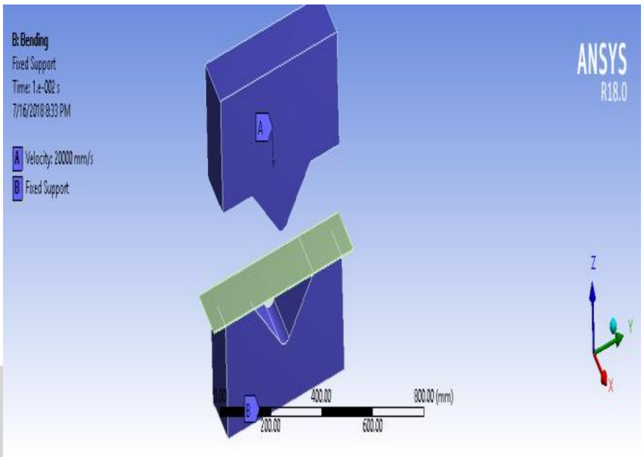


Fig2. Model in Ansys Work Bench

After creating the model it is imported in Explicit Dynamics application of Ansys Workbench. Then the boundary conditions are applied to the model. The die is fixed and doesn't move; hence fixed support is given to die. The punch is movable and hence initial velocity is given as 20m/s to it in the downward direction (towards die).

Contacts- There is frictionless contact in between lower surface of sheet and the surface of die cavity. So as the punch approaches towards the die the sheet flows in the gap between the die and punch but remains in contact with the die cavity.

3. **Meshing:**

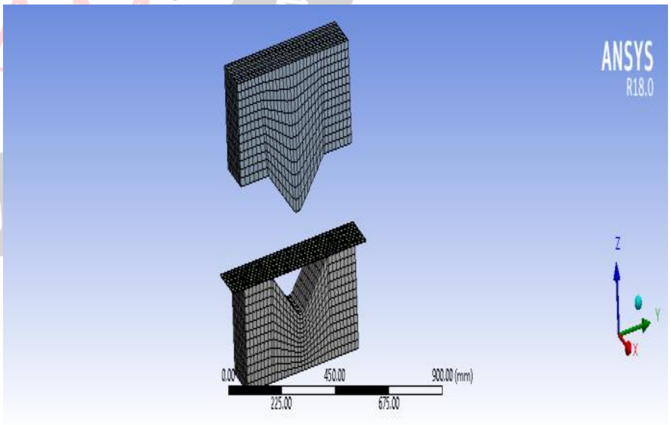


Fig 3. Meshing in Ansys Work Bench

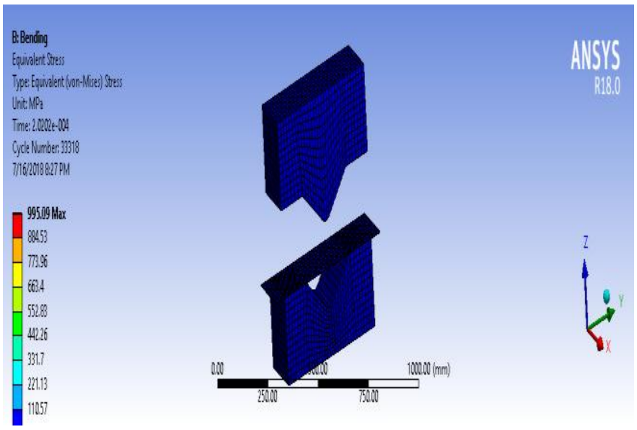
The model is then meshed with fine meshing. All the elements are 3D brick type8 node elements. The total number of elements generated is 7317 and the total number of nodes generated is 9390.

Analysis setting- Maximum number of cycles are 10, 00,000 and the analysis end time specified is 0.01 sec., maximum energy error is specified as 1%

 DOI : 10.18231/2454-9150.2018.0559

ISSN : 2454-9150 Vol-04, Issue-04, July 2018

4. Results:



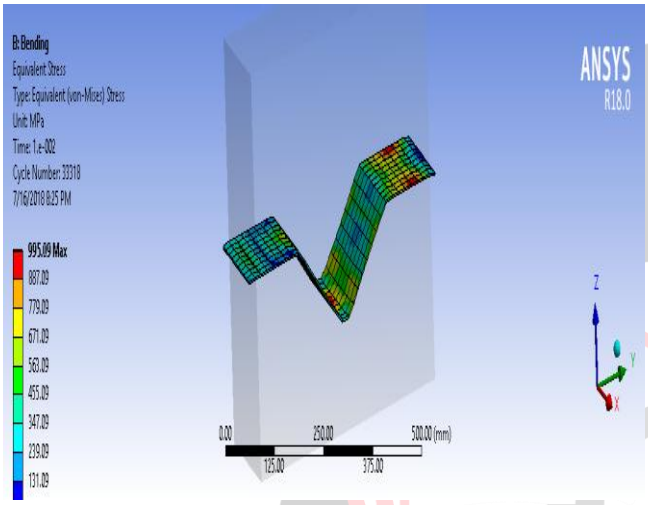
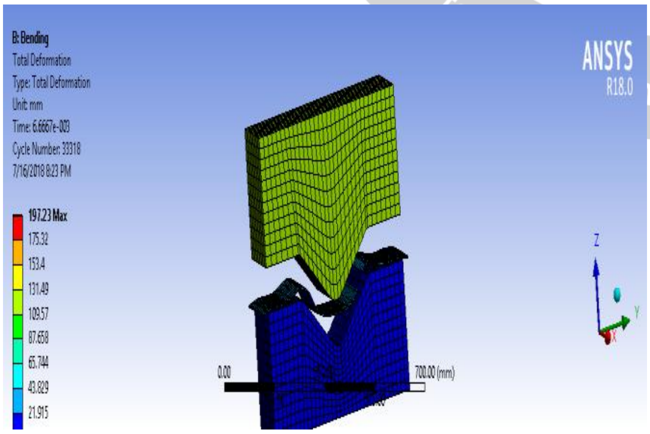


Fig 4. Analysis Results

The set-up is solved for the given conditions and the results are obtained. In the above figure, the total equivalent stress (Von-Mises) is shown. The total stress found is 995 Mpa for the given speed of the punch. Maximum stress found is near middle bend of the sheet as highest force is at the tip of the punch.



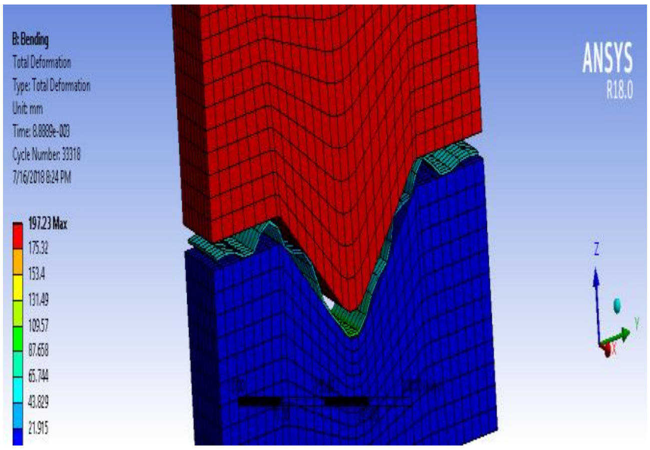


Fig 5. Analysis Results

In the above figure total deformation shown is 197 mm, which is total distance traveled by the punch.

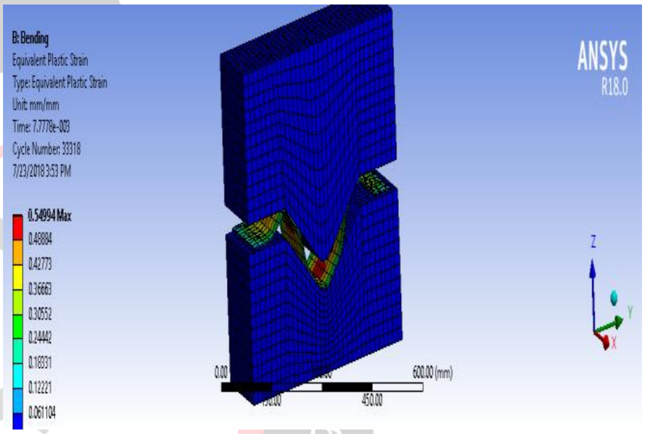


Fig 6. Analysis Results

The total plastic strain is shown in above figure, which is found to be 0.55 mm/mm.

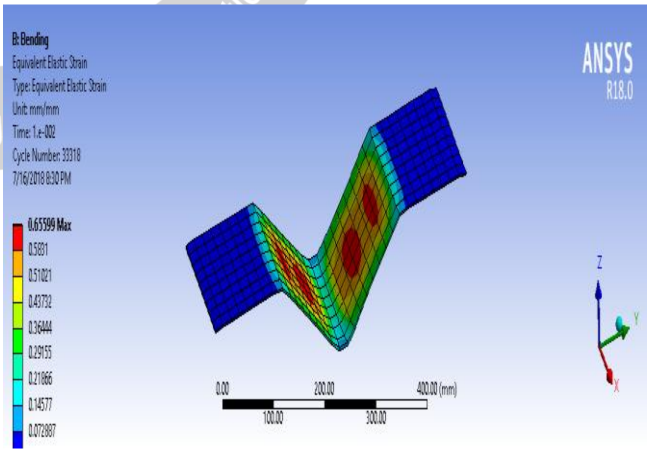


Fig 6. Analysis Results

The total elastic strain is found to be 0.65 mm/mm which is shown in the above figure. The elastic strain is observed slightly more than the plastic strain.

 DOI : 10.18231/2454-9150.2018.0559

VI. Conclusion

From above work following conclusions are made

1. The total stress found 995 Mpa is more than the yield strength of the material, hence the sheet deforms into the shape of the die cavity.
2. The elastic and plastic strain observed in the analysis is within limit i.e. (0.65 & 0.55) which is less than 1 mm/mm hence the analysis is correct.

References

[1] Kalpakjian, Serope; Schmid, Steven, "Manufacturing Engineering and Technology", (International edition. 4th ed.). Prentice Hall. ISBN 0-13-017440-8.2001.

[2] Shambhuraje Jagatap, Prof. Bharat S. Kodli, "A Finite Elemental Study of Contact Pressure Distribution in Stamping Operations", Vol. 2 Issue 10, October – 2013, International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 10, ISSN: 2278-0181, 2013

[3] Trivikram. N L, 1Vasantha R Sural, 1Mrityunjaya R Yeli, 1Ramesh Venkatesan, 2Dr.Ramesh.C.S, "stamping simulation study on al-6061 alloy using fea approach,", 3rd ANSA & μETA International Conference September 9-11, 2009 Olympic Convention Centre, Porto Carras Grand Resort Hotel, Halkidiki Greece. 2009.

[4] Xiaoli Fu,Yuedong Qiu,Mingtu Zhao, "Finite Element Analysis of Stamping of Automobile Beam", Journal of Networking and Telecomunications,

[5] Aminu Saleh Mohammed, "The Hydrodroforming Process of an Automotive Part", International Journal of Engineering Research and Development e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com Volume 8, Issue 7 (September 2013), PP. 49-55

[6] Pradeep Mohan, Vinay Nagabhushana, Cing-Dao (Steve), KanJon Riley, 5. LS-DYNA Anwenderforum, Ulm 2006, https://pdfs.semanticscholar.org/0ddc/32a053a31ba2f9a864d60581192f4095f0fd.pdf

[7] Susie Bozzini, Donald Friedman, Raphael Grzebieta, "Vehicle Roof Structure Design Can Significantly Reduce Occupant Injury", Proceeding of International Crashworthiness Conference 2014.

[8] Joachim Scheub, Fadi Tahan, Kennerly Digges, Cing- Dao Kan, "Influence of Different Platen Angles and Selected Roof Header Reinforcements on the Quasi- Static Roof Strength of a 2003 Ford Explorer FE Model", IRCOBI Conference 2013.

[9] Keith Simmonsa, Mario Mongiardinia , Raphael Grzebietaa , George Rechnitzera, "Effect of Vehicle Roof Shape on Rollover Safety", Proceedings of the 2015 Australasian Road Safety Conference 14 – 16 October, Gold Coast, Australia.

[10] Eriksson,D.: Testing and evaluation of material data for analysis of forming and hardening of boron steel components: Modeling and simulation in material science and engineering (2002), pp. 277-294

[11] Alexander Muizemnek and Konstantin Zhekov, "Cap Shaped part forming simulation", CAD-FEM GmbH Representation in CIS, Moscow. Bradley N. Maker and Xinhai Zhu, "Procedure for spring back analysis using LSDyna",LSTC Publications, October 2003.

[12] Alexander.J.M and Price J.W.H, "Finite Element analysis of hot metal forming", 18th Int. Machine Tools des. Res. Conf., pp.267-74 (1977)

[13] Baaijens F.P.T , Veldpaus F.E and Brekelmans W.A.M " Numerical simulations of contact problems in forming process" , 2nd Int.Conf..on numerical in Industrial forming processes,Balkema Press,pp 85-90

[14] Chen C.C and Koboyashi.S, "Rigid plastic finite element analysis of ring compression applications of numerical methods to forming processes." ASME, AMD, 28th Conf. pp.163-74 (1978).

[15] S. Thipprakmas, M. Jin, K. Tomokazu, Y. Katsuhiro, M. Murakawa, Prediction of fine blanked surface characteristics using the finite element method (FEM), Journal of Materials Processing Technology 198 (2008) 391-398.

[16] Haber R.B and Hariandia B.H " An eulerian-lagrangian finite element approach to large-deformation frictional contact ", Comput .Structure Journal.20,P.No 193-201

 DOI : 10.18231/2454-9150.2018.0559