iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 Issue: VII Month of publication: July 2018

DOI:

Finite Element Analysis of Automotive Roof Header Manufactured by Stamping Process

Kete Krishnaraj Shivaji[1], Prof. A.V. Chipade[2]
[1]M.E. Student, Department of Mechanical Engineering, BMIT, Solapur
[2]Head of Department, Department of Mechanical Engineering, BMIT, Solapur

***Abstract:** Roof header made of Al-6061 alloys an important component in a passenger car. Roof headers profile 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. 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. 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

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 analysing the contact pressure distribution directly at the tooling-work piece interface [1-5].

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 [6]. 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.

In the present work, two types of analysis are considered. One three dimensional symmetric object is analysed 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 [7-8].

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-misses, 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 [9, 10].

II. LITERATURE REVIEW

The goal of this study was to determine the detailed design of a greenhouse structure (roof and pillars), such that when it is loaded in a static roof crush test th0e force-displacement response mimics that of a modern full-size crossover vehicle. This study was carried out using finite element analysis with the goal of identifying a specific design to be fabricated for use [11].

Stamping is very important in aspect for manufacturing of any sheet metal components. In the past decades it has been found that stamping simulation can save lot of metal and also can be cost effective, will make the manufacturing cheaper and with better quality product formed. The strain and von-Mises stress plays a major role in the forming process. Many times it's possible to get the risk of cracks in the locations near the transition from concave and convex geometry [12].

A comparative study based on three different parameters and we along with results based on the combination of the parameters tabulated. Following are the parameters considered for the stamping. Variation of Blank Thickness, Variation of Clamp Holding Pressure, Variation of Ram Velocity.

The simulation of stamping is rather complex and challenging, and requires information about properties of die and sheet materials and friction between the deforming material and the dies. The computer simulation is very popular and it is based on the finite element method, which allows to better understand the interdependence between parameters of process and choosing optimal solution [13, 14].

Optimizing the stamping process for the metals in producing high quality components is time consuming and very expensive because of much Iteration involving several combinations of the designs of the die and the punch and the work piece materials, In the light of the above: researchers are currently adopting FEA based software's to overcome the challenges associated with the stamping process. With the advent of high computing technologies, complex shaped sheet metal profiles for automotive applications can be modelled with ease and can be imported to FEA solvers for design optimization studies of die and tooling [15, 16].

III. SCOPE AND OBJECTIVE OF WORK

A. Scope of the Work

A review of published literature reveals that very few studies have been made on application of simulation software in analysis of automobile roof header which is manufactured by stamping operation. 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.

B. Objective of the Work

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. The results are captured for von-misses, 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.

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

A. Initial built up of geometry of movable die, fixed die and sheet metal
B. Meshing with three dimensional elements
C. Contact pair creation between fixed die, movable die and sheet metal
D. Study of Material Properties.
E. Applying Boundary Conditions such as Punch velocity, holder force.
F. Solving the problem with different temperature dependent material data.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VII, July 2018- Available at www.ijraset.com

G. Analysing the problem
H. Results presentation

V. FINITE ELEMENT 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.

A. Model

The model is created in Design Modeller which is the application of Ansys. The model consists of a die, a punch and a sheet which is to be deformed.

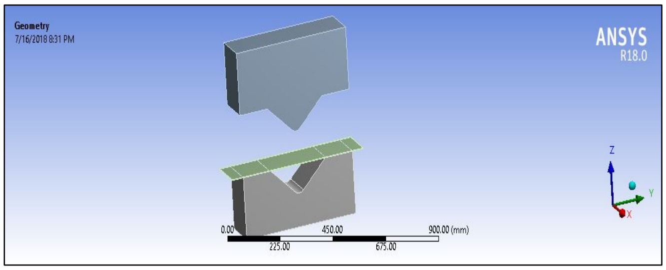


Fig1. Model in Design Modeller

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.

B. Boundary Conditions

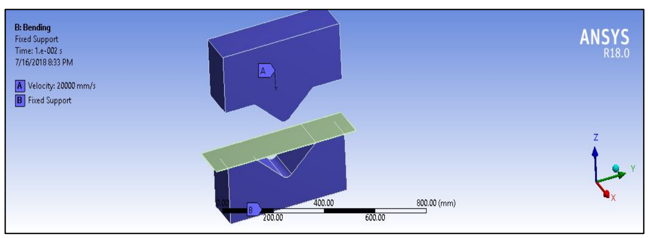


Fig2. Model in Ansys Work Bench

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.

C. Meshing

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%

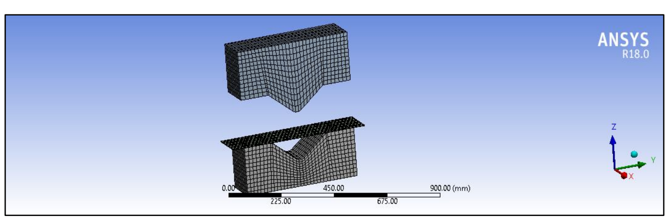


Fig 3. Meshing in Ansys Work Bench

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VII, July 2018- Available at www.ijraset.com

D. Results

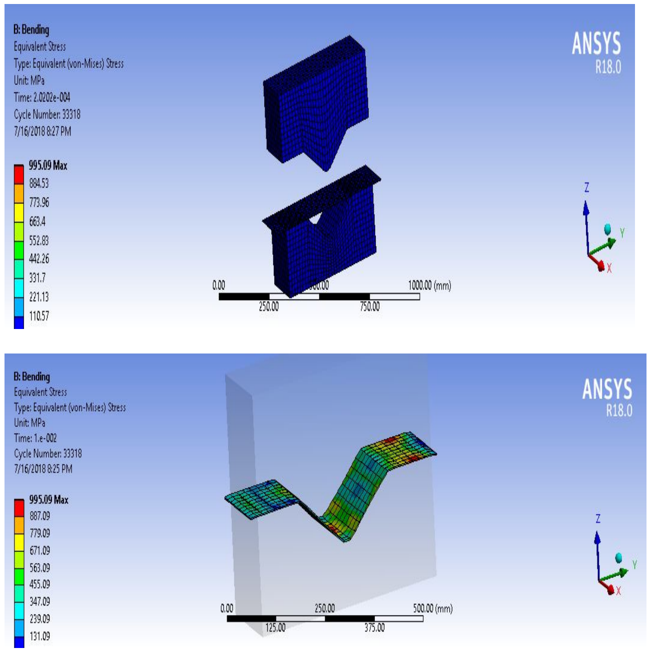


Fig 4. Analysis Results

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.

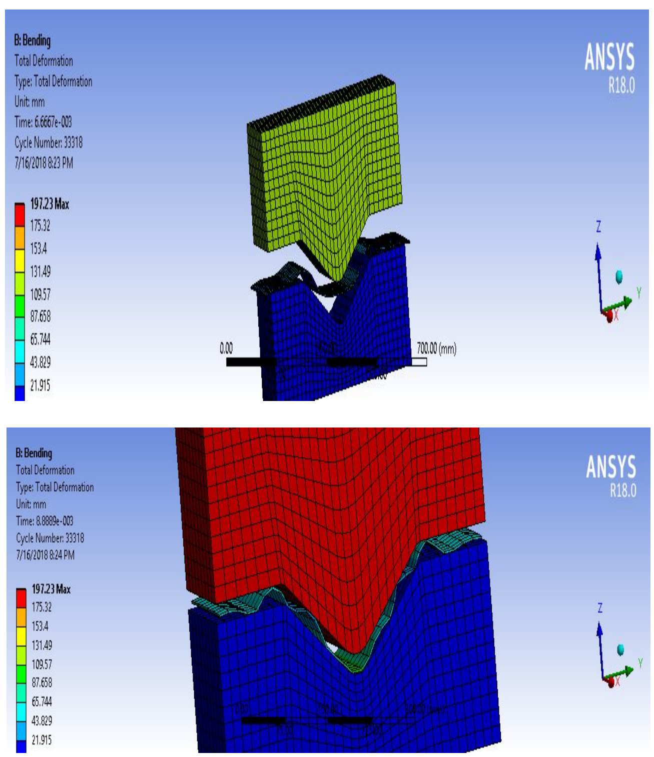


Fig 5. Analysis Results

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

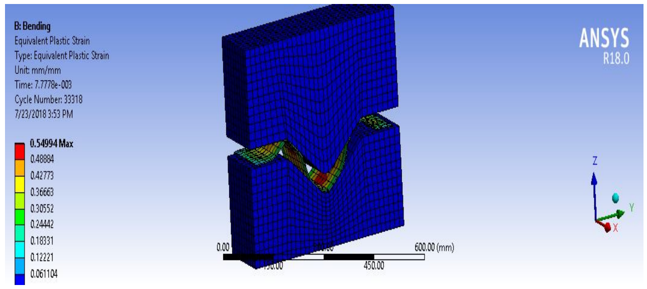


Fig 6. Analysis Results

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

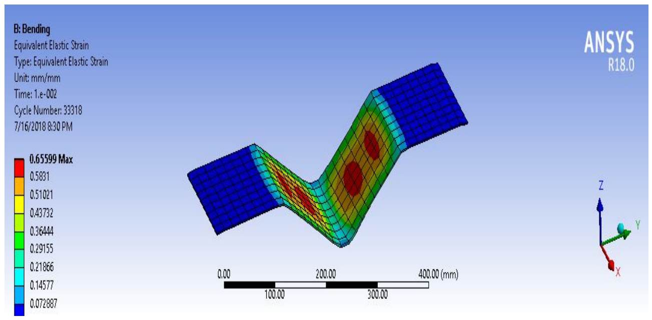


Fig 6. Analysis Results

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.

VI. CONCLUSIONS

From above work following conclusions are made

A. 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.

B. 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.

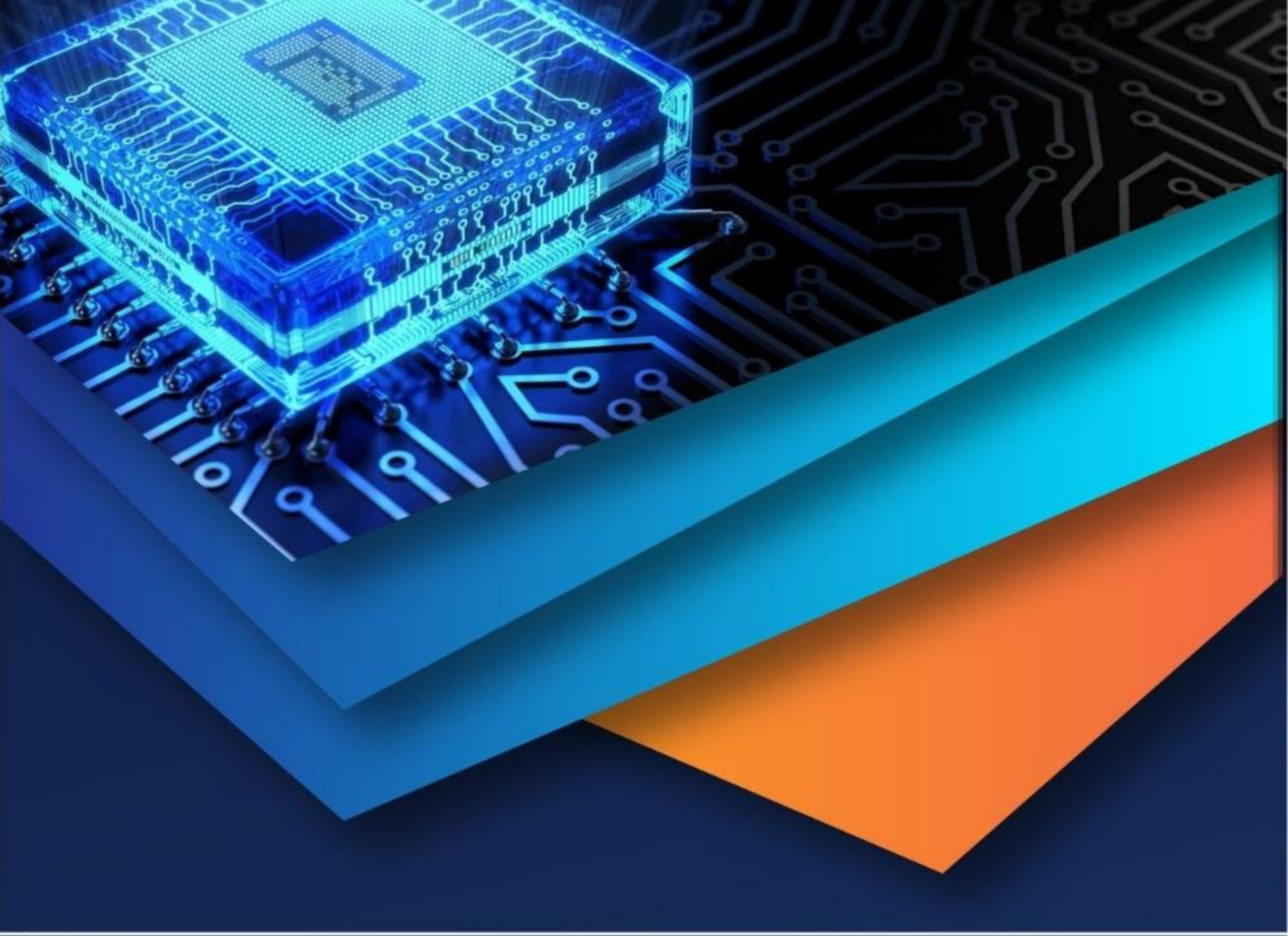
[12] Alexander.J.M and Price J.W.H, "Finite Element analysis of hot metal forming", 18th Int. Machine Tools des. Res. Conf., pp.267-74 (1977)

[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



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY