A Review on Finite Element Analysis of Automobile roof header Manufactured By Stamping Process

Mr. Kete Krishnaraj Shivaji

M.E. (Mech) Student/Department of mechanical engineering

BRAHMDEVDADA MANE INSTITUTE OF TECHNOLOGY, SOLAPUR

Prof. A.V. Chipade

Department of mechanical engineering

BRAHMDEVDADA MANE INSTITUTE OF TECHNOLOGY, SOLAPUR

Abstract

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. 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.

Keywords— stamping operations, pressure distribution, hydraulic or mechanical press.

Introduction

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 vonmises, 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.

LITERATURE REVIEW

Eriksson.D.[1]: 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.

Alexander Muizemnek and Konstantin Zhekov, [2]   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.

Alexander.J.M and Price J.W.H,[3] 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.

Chen C.C and Koboyashi.S,[4] 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. 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 modeled with ease and can be imported to FEA solvers for design optimization studies of die and toolings.

S. Thipprakmas, M. Jin, K. Tomokazu, Y. Katsuhiro, M. Murakawa[5], 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 modeled with ease and can be imported to FEA solvers for design optimization studies of die and tooling's.

RESEARCH METHODOLOGY

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

OBJECTIVES

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

CONCLUSION

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. 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.

References

[1] 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

[2] 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.

[3] 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)

[4] 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).

[5]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.