International Journal of Scientific Research and Management Studies (IJSRMS)
ISSN: 2349-3771 Volume 2 Issue 7, pg: 288-293

DEPLOYING CAE FOR 'FORMING ANALYSIS' OF A SHEET-METAL AUTOMOTIVE COMPONENTS WHILE VALIDATING THROUGH PHYSICAL EXPERIMENTATION

[1]Asad Aslam Mujawar, [2]Vijay J. Khot, [3]Swapnil S. Kulkarni
[1]ME (Mech-Design), [2]Asst. Professor,
Dr. J J Magdum College of Engineering, Jaysingpur. Shivaji University, Kolhapur, India
[3]Associate, Advent Tool Tech, Bhosari, Pune, India

ABSTRACT

Sheet metal parts involve cutting and non-cutting operations. This work shall focus on 'forming' as a non-cutting operation that accounts for a significant volume of parts processed using this operation. Forming, being a tricky operation to deal with, is best approached using CAE solvers. The behavior of the material during this operation depends upon the material properties and is manifested by the Forming Limit Diagram Curve. The feature like radius on the die-block, velocity of ram, use of lubricant, blank holding pressure, etc plays a key role in delivering a defect-free component. Hyper Form shall be deployed as a solver for the given problem case. Mathematical tools shall be used for preliminary investigation for finding tonnage. Validation shall be realized using physical experimentation during trials or testing.

KEYWORDS: *Forming, Draw, FLD, Hyper Form, Tonnage.*

I. INTRODUCTION

Forming processes are particular manufacturing processes which make use of suitable stresses (like compression, tension, shear or combined stresses) to cause plastic deformation of the materials to produce required shapes. In sheet metal forming, the final shape of a part is made from a flat metal sheet. The desired shape is achieved through plastic deformation, without undergoing any machining like milling. In most cases, a certain amount of elastic deformation leads to springback which occurs after forming is complete.

Most Automotive parts are made up of sheet metal which is manufactured by using Forming processes. Different type of reinforcements, body parts and door parts are manufactured in sheet metal scope. Thus sheet metal forming plays a very important role in automotive industry. With every manufacturing process there are some defects associated with it. In forming process also there are some types of defects arises, but the most common defects related to automotive body parts are: Springback, wrinkles, tearing, thinning. These defects have many cost effective impacts like Material loss, loss of productivity, rejection, rework, Quality issues, etc. To avoid these losses analysis of forming processes is important. The feature like radius on the die-block, velocity of ram, use of lubricant, blank holding pressure, etc plays a key role in delivering a defect-free component.

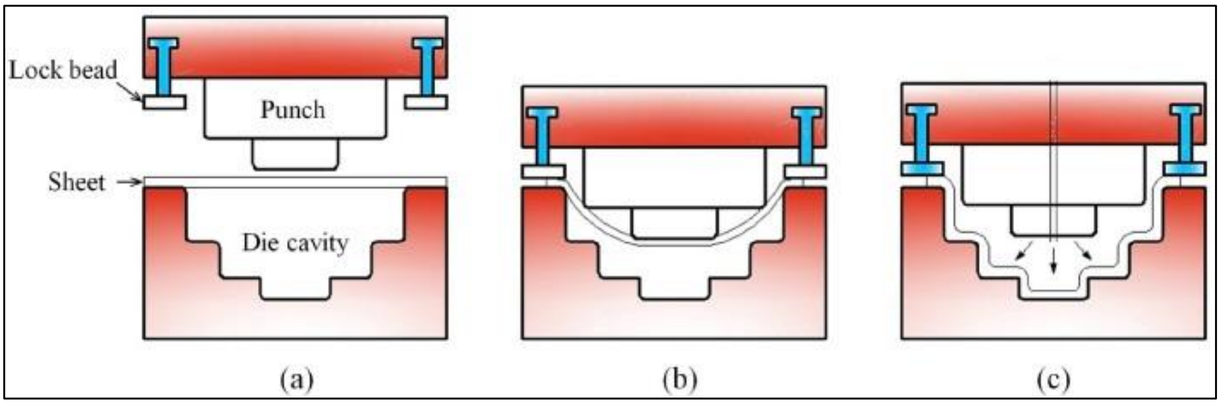

Fig.1: Sheet metal forming Process

II. LITERATURE REVIEW

The first effort at simulating metal forming was made using the finite difference method in the 1960s to better understand the forming process, later on further study happen to optimize the parameters in die design to minimize the defect at production station stage, which is breakthrough for saving valuable time saving in design and development.

Mentioned journal paper from different sources is research work in the field of forming analysis to minimize defect.

Seong-Chan Heoet al. [1], has study on thick plate forming using flexible forming process and it's applied to a simply curved plate in prototyping. Prior to manufacturing of a prototype, numerical simulations for a saddle-typed thick plate forming process including metal forming and spring-back analyses are carried out to predict the forming performance. Experiments carried out of flexible forming process. In the simulation, flexible forming process and spring-back analyses are carried out and the configuration of the results is compared with the prototype with respect to curvature radii. It is confirmed that the flexible forming process and its forming machine are appropriately designed.

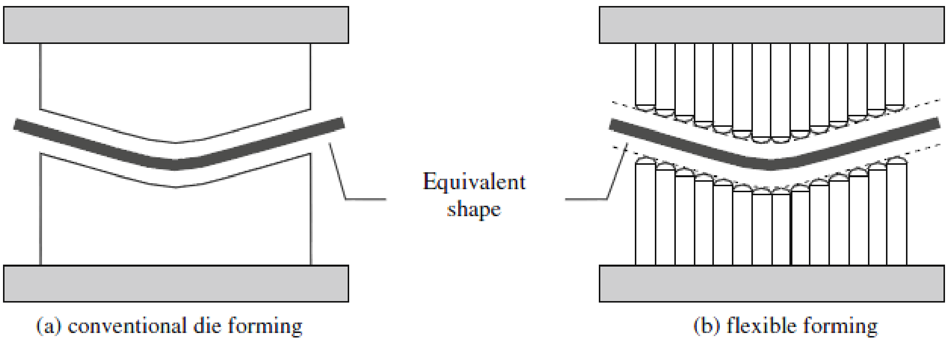

Fig.2: Die and Flexible forming process

Giovanni B. Broggiatoet al. [2], explains about Computer-aided engineering for sheet metal forming process and define spring back quality function for robust design process. For advanced high strength steels, spring back reduction is required. An approximated formulation to compute the springback field after stamping through the finite element analysis of the process. This can be found assuming that the residual field of nodal forces after stamping produces a springback shape referable to a linear combination of n modes of vibration of the nominal shape of the component. At an early stage of component design, the finite element code adopted is usually based on the inverse approach (or one-step approach). By this kind of FEA, a fast evaluation of the stamping force range and an overview of the general formability of the component are possible. That helps in the next die design definition, regulating the proper set up of punch and die shape or the product restyling of the most critical details.

Eugenio Oñate et al. [3] used a particle finite element method for analysis of industrial forming Processes. There is a large number of forming manufacturing processes in industry that involve the interaction of highly deformable continua, including viscous fluids and solids that undergo large deformations. Lagrangian formulation uses for analysis of industrial forming processes involving thermally coupled interactions between deformable continua. The governing equations for the deformable bodies are written in a unified manner that holds both for fluids and solids. A residual-based expression of the mass conservation equation obtained using the FIC method provides the necessary stability for quasi/fully incompressible situations. The governing equations for the generalized continuum are discretized with the FEM using simplicial elements with equal linear interpolation for the velocities, the pressure and the temperature. The merits of the formulation in terms of its general applicability have been demonstrated in the solution of a variety of thermally-coupled industrial forming processes.
D. Y. KIMet al. [4], estimate the life of hot press forming die by using interface heat transfer coefficient obtained from inverse analysis. During the hot press forming process, the die experiences repeated thermal and mechanical loads owing to exposure to the heated workpiece. Such repeated thermal loads may cause deterioration in the mechanical properties of the die, leading to fatigue failure. Inverse finite element method (FEM) analysis of the hot press forming process was performed to determine the interface heat transfer coefficient. The interface heat transfer coefficient was applied to the FEM simulation, and the temperature distribution and stress values for the die were determined. Considering the thermo-mechanical stress history, the fatigue life of the die was estimated based on the stress-life approach. Comparison of the die stress magnitudes for the hot and cold press forming conditions confirmed that the die life may be significantly reduced under hot press forming conditions. It concludes that accurate data of the interface heat transfer coefficient are indispensable for reliable prediction of die life.
Seong-Chan Heoet al. [5], Study and evaluate Shape error compensation in flexible forming process using over bending surface method. The design of the flexible forming process considering die shape compensation using an iterative over bending method based on numerical simulation is carried out. In this method, the springback shape obtained from the final step of the first forming simulation is compared with the desired objective shape, and the shape error is calculated as a vector norm with three-dimensional coordinates. The error vector is inversely added to the objective surface to compensate both the upper and lower flexible die configuration. The configuration of the prototype obtained from the experiment is compared with the numerical simulation results, which have consideration of the springback compensation. Consequently, it is confirmed that the suggested method for compensating the forming error could be used in the design of the flexible forming process for thick-curved plates.
Thomas Masséet al. [6], study and analyze Mechanical and damage analysis along a flat-rolled wire cold forming schedule. Numerical simulation is used to study patented high-C steel flat-rolled wire cold forming processes. An elasto-plastic power law, identified from mechanical tests, is used into Forge2005® finite element (FEM) package in order to describe the material behavior during wire drawing followed by cold rolling. The experimental (SEM) analyses [22] have shown that damage is initiated at wire-drawing stage and fully develops during rolling. The final state is a myriad of small cavities everywhere in the wire, although more concentrated in the core area.

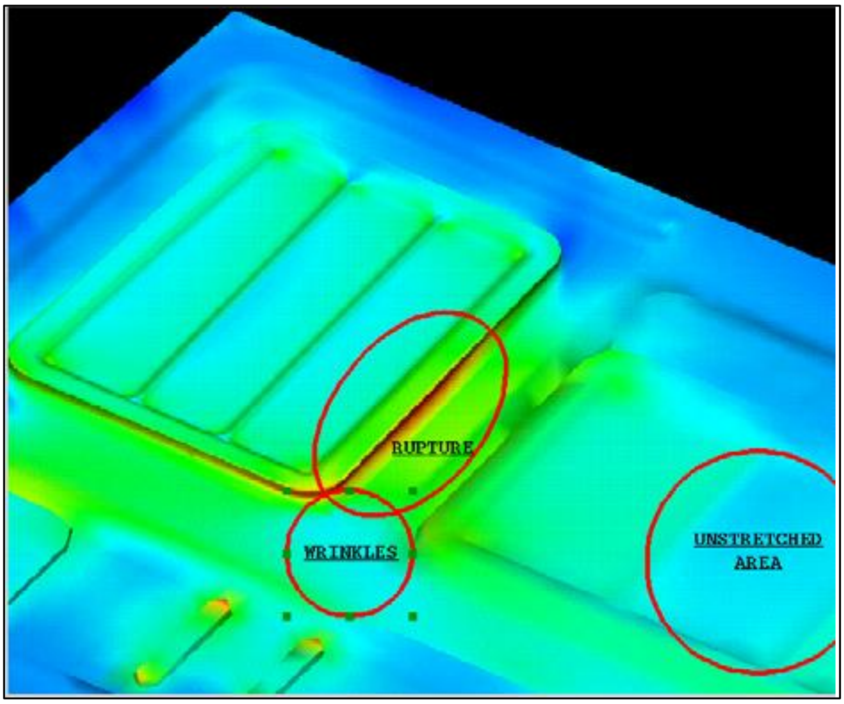

Fig.3: Defects in forming process

b. Methodology:

1. Computational Method-

1. Blank development using CAD Software such as CATIA V5, UG-NX and Creo-3.
2. Pre-processing or die-punch setup using HyperMesh Interface
3. Applying loads and boundary conditions
4. Solving using HyperForm as a Solver
5. Post-processing using HyperView interface
6. Generating alternative/s while altering processing parameters
7. Repeat step no. 4 & 5
8. Recommend the best alternative

2. Numerical Method-

In numerical method, we shall attempt to conduct preliminary investigation for the problem through calculation of the required tonnage for forming the component. Using industrial practices for assigning blank holding force, selection of spring, bush, guide-pillar and press selection as shall be suitable for the component and the operation.

3. Experimentation-

Experiments shall be conducted on a press of a suitable type and capacity. The die would be mounted on the bolster plate of the press and the speed of the ram would be set based on the historical data as well as the input received from the analysis data (simulation).

Forming problems can be predicted before tool fabrication through the use of software that can be integrated into production routes which rely increasingly on computer technology. The prediction of forming difficulties at the component design stage ensures that the chosen geometry is compatible with the formability of steel. Forming has become a highly technical process, and the development of a steel forming route no longer involves simple trial and error methods. Close collaboration between component designers, forming engineers and steelmakers guarantees the industrial feasibility of new parts with very short development times associated with the Experimentation/ Tryout phase.

The parameters influencing the `form' operation during the trials could be listed as:

- Type of material (influencing the Limiting Draw Ratio)
- Thickness of the component
- Blank holding pressure

- Speed of the operation

For this work, the critical parameter/s (one or two of the above) shall be identified and modified to realize a desired response.

III. CONCLUSION/ EXPECTED RESULTS

- Computational methodology seems to be the best option to be followed into the criticality of the geometry of the part and the forming operation
- HyperForm has been identified to be a suitable solver for pursuing the `computational' results
- Physical experimentation with one significant factor could be considered for validating the work

IV. FUTURE SCOPE

1. More control parameters like use of lubricant could be explored for future research work
2. Alternative software for computational techniques can be adapted based on the support from Industry
3. Alternative type of press (Mechanical or Hydraulic) could be deployed to evaluate its influence on the process
4. Factors like Blank holding pressure could be considered for study for assigning a precise level (value)
5. Provision of beads based on suitability for the given operation could be explored further

REFERENCES

[1]. Chan Heo, Young-Ho Seo, Tae-Wan Ku and Beom-Soo Kang, **“**A study on thick plate forming using flexible forming processand its application to a simply curved plate” Seong, Springer-Verlag London Limited 2010, Issued on 31 March 2010.

[2]. Giovanni B. Broggiato, Francesca Campana and Edoardo Mancini, “Computer-aided engineering for sheet metal forming process and define spring back quality function for robust design process”, Springer-Verlag London Limited 2012, Issued on5 September 2012.

[3]. Eugenio Oñate, Alessandro Franci and Josep M. Carbonell,“A particle finite element method for analysis of industrial formingProcesses” Springer-Verlag Berlin Heidelberg 2014, Issued on 17 April 2014.

[4]. D. Y. KIM, H. Y. KIM, S. H. LEE and H. K. KIM, “Life estimation the life of hot press forming die by using interface heat transfer coefficient obtained from inverse analysis”, Department of Automotive Engineering, Kookmin University, Seoul 136-702, Korea, Issued on 8 May 2014.

[5]. Seong-Chan Heo, Jae-Nam Kim, Woo-Jin Song, Tae-Wan Ku and Beom-Soo Kang, “Shape error compensation in flexible forming process using over bending surface method”, Springer-Verlag London Limited 2011, Issued on 20 August 2011.

[6]. Massé, YvanChastel, Pierre Montmitonnet, Christian Bobadilla, Nicolas Persem and Sylvain Foissey “Mechanical and damage analysis along a flat-rolled wire cold forming schedule” Thomas, Springer-Verlag France 2011, Issued on 26 February 2011.

[7]. Taylan Altan, Erman Tekkaya, “Sheet Metal Forming: Processes and Applications,”, Chapter 3: Process Simulation,”, August 31, 2012.

[8]. Winfried Schmitt, Oleg Benevolenski, Tom Walde, Andriy Krasowsky,“Material Characterization for Simulation of Sheet Metal Forming”, VIII International Conference on Computational Plasticity (COMPLAS VIII), Barcelona, 2005.

BIOGRAPHY

Asad Aslam Mujawar, M.E. Machine design, Department of Mechanical Engineering, Dr. J J Magdum College Of Engineering, Jaysingpur. Shivaji University, Kolhapur.

Vijay J. Khot, Asst. Professor, Department of Mechanical Engineering, **J J Magdum** College Of Engineering, Jaysingpur. Shivaji University, Kolhapur.

Swapnil S Kulkarni Associate, Advent Tool Tech India Pvt. Ltd., Pune. The Company offers Engineering Services and Manufacturing Solutions to Automotive OEM's and Tier I and Tier II Companies. He is a Graduate in Industrial Engineering with PG in Operations Management. With around 20 years of working experience in the domain of R&D, Product Design and Tool Engineering, he has executed projects in the Automotive, Medical and Lighting Industry. His area of interest is Research and Development in the Engineering Industry as well as the emerging sector of Renewable Energy.