Global Journal of Researches in Engineering
Automotive Engineering
Volume 13 Issue 1 Version 1.0 Year 2013
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN: 2249-4596 & Print ISSN: 0975-5861

Sheet Metal Forming Simulations for Heavy Commercial Vehicle Parts by LS-DYNA

By Abdulla Mohammad Gous Shaikh & Tippa Bhimasankara Rao
Institute of Science

Abstract - The design of new forming tools gets more proble-matic as the geometries get more complicated and the materials less formable. The idea with this project is to evaluate if an implementation of simulation software in the designing process, to simulate the forming process before actually building the tools, could help to avoid expensive mistakes. To evaluate this, the commercial FEM simulation software LS-DYNA was used in a complicated project, where the design of the forming tools for forming a girder was considered. The main objective was to avoid cracking and severe wrinkling which may result in the forming process. With help of simulations a stable forming process which did not yield cracks or severe wrinkling, was eventually found. The girder was almost impossible to form without cracking, but the breakthrough came when we tried to simulate a performing step which solved the problem. Without simulation software this would never have been tested since it would be too risky and expensive to try an idea which could turn out to be of no use. The simulations also showed that the spring back - shape deformation occurring after pressing - was large and hard to predict without simulations. Therefore, the tools were also finally spring back compensated.

Keywords : forming, forming limit diagram (FLD),LS-DYNA, spring back, stamping, stress, strain.

GJRE-B Classification : FOR Code: 090299

SHEET METAL FORMING SIMULATIONS FOR HEAVY COMMERCIAL VEHICLE PARTS BY LS-DYNA

Sheet Metal Forming Simulations for Heavy Commercial Vehicle Parts by LS-DYNA

Abdulla Mohammad Gous Shaikh [α] & TippaBhimasankara Rao [σ]

Abstract - The design of new forming tools gets more problematic as the geometries get more complicated and the materials less formable. The idea with this project is to evaluate if an implementation of simulation software in the designing process, to simulate the forming process before actually building the tools, could help to avoid expensive mistakes. To evaluate this, the commercial FEM simulation software LS-DYNA was used in a complicated project, where the design of the forming tools for forming a girder was considered. The main objective was to avoid cracking and severe wrinkling which may result in the forming process. With help of simulations a stable forming process which did not yield cracks or severe wrinkling, was eventually found. The girder was almost impossible to form without cracking, but the breakthrough came when we tried to simulate a performing step which solved the problem. Without simulation software this would never have been tested since it would be too risky and expensive to try an idea which could turn out to be of no use. The simulations also showed that the spring back - shape deformation occurring after pressing - was large and hard to predict without simulations. Therefore, the tools were also finally spring back compensated.

We concluded that simulations are very effective to quickly test new ideas which may be necessary when designing the tools for forming complicated parts. Simulation also provided detailed quantitative information about the expected cracks, wrinkles, and weaknesses of the resulting pieces. Even though there is cost associated with simulations, it is obvious from this project that simulation software is a it is actually very less as compared to actual tool building and making mistakes.

Keywords : forming, forming limit diagram (FLD), LS-DYNA, spring back, stamping, stress, strain.

I. Introduction

a) The forming process

Sheet metal forming, or stamping, is a process where a material, referred to as the blank, is formed by stretching it between a punch and a die, see figure (1). First, the punch and the binder or blank-holder is in an uplifted position. The binder then moves down and clamps the blank in such a way that the blank is allowed to be drawn inwards, but still creates tension to stretch the blank between the die and the punch. In somecases, draw-beads are also used to further increase the resistance of inwards drawing. After this, the punch moves down and the sheet is drawn through the opening of the die ring.

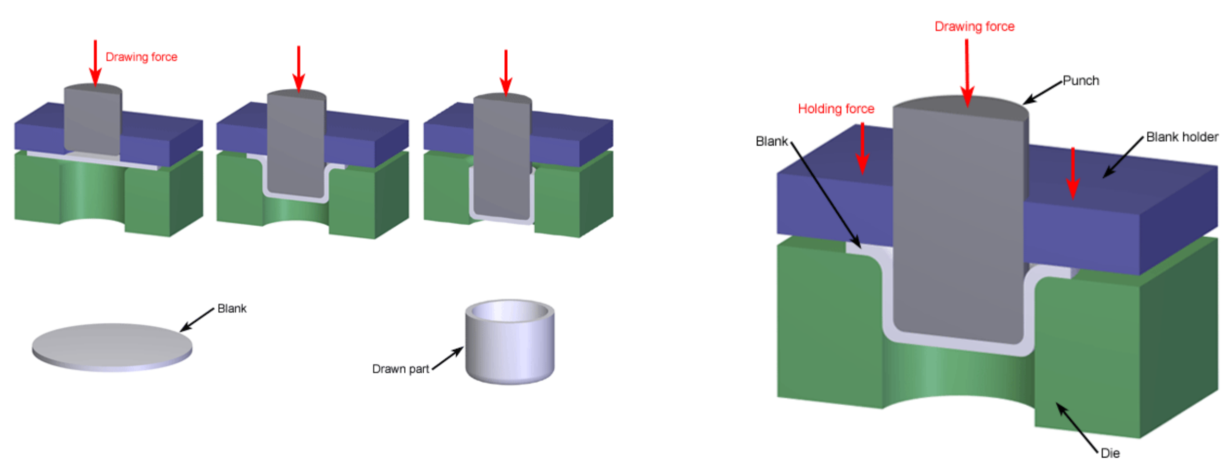


Figure 1

Note that the forming process is not a compression process, but rather a stretching process, i.e. the blank is stretched over the tools. Since the blank, for the most part, only is in contact with the tooling on one side, no through-thickness compression is present. The flow stress in the sheet is generally much larger than the contact pressure, and thus, it is assumed that there is no through-thickness compression. This is called plane stress deformation. Stamping or forming

Author α : PG Student, Department of Mechanical Engineering, Nimra Institute of Science and Technology Vijayawada, AP, India.
Author σ : HOD, Department of Mechanical Engineering, Nimra Institute of Science and Technology, Vijayawada, AP, India.

can be done in a single or double-acting press. A double-acting press is a press in which the clamping and punch actions are separate. Usually the forming has to be done in several stages, different presses, if the desired geometry is complex.

II. Mechanics Of Sheet Metal Forming

Formability is the ability of a given metal work piece to undergo plastic deformation without being damaged. The plastic deformation capacity of metallic materials, however, is limited to a certain extent, at which point, the material could experience tearing or fracture (breakage).

Sheet metal parts are produced in large quantities using special tooling and high-volume production techniques. Most processes are tensile in nature and deformation in one stage is limited by the onset of tensile instability, necking and tearing. Because of the compressive stress in some part of thin sheet membrane, wrinkling and buckling may also occur. This makes very important to understand the mechanics of the forming process, knowledge of failure analysis and considerable experience in manufacturing.

The true stress-strain curve as shown in fig 2. Derived from a tensile test can be applied to other deformation processes that may occur in typical sheet-forming operations. A common feature of many sheet-forming processes is that compared with the stresses in the plane of the sheet (the membrane stresses), the stress perpendicular to the surface of the sheet is small. If this normal stress is assumed zero, a major simplification is possible. Such a process is called plane stress deformation.

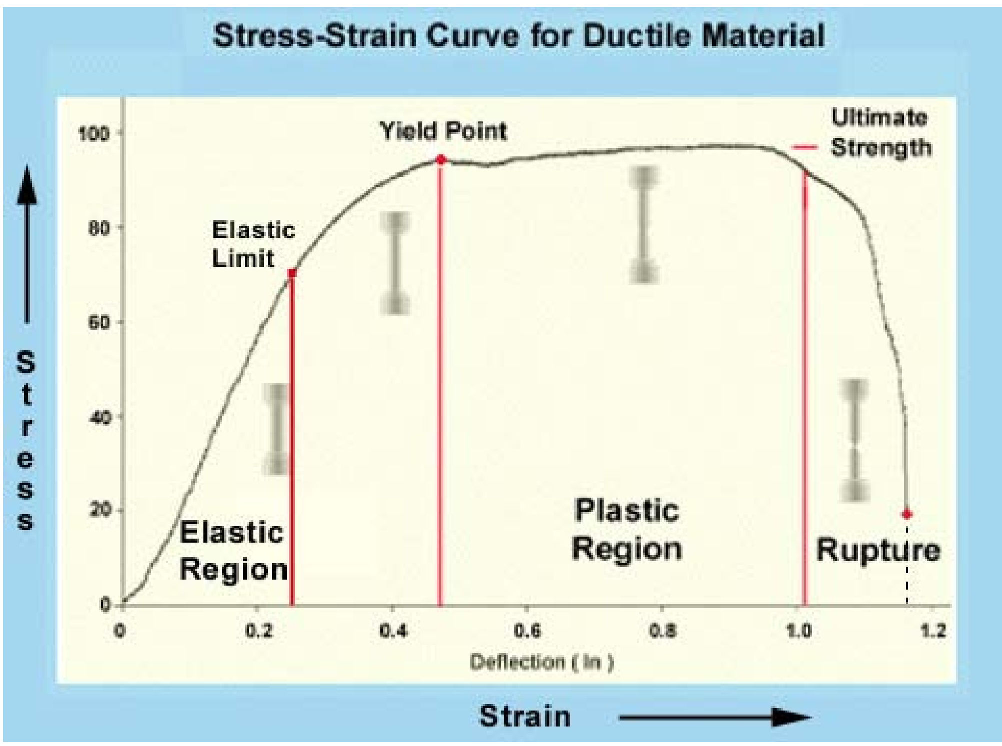


Figure 2

a) Forming-Limit Diagrams (FLD)

A forming limit diagram, also known as a forming limit curve, is used in sheet metal forming for predicting forming behaviour of sheet metal. The diagram attempts to provide a graphical description of material failure tests, such as a punched dome test.

In order to determine whether a given region has failed, a mechanical test is performed. The mechanical test is performed by placing a circular markon the work piece prior to deformation and then measuring the post-deformation ellipse that is generated from the action on this circle. By repeating the mechanical test to generate a range of stress states, the formability limit diagram can be generated as a line at which failure is onset.

Sheet metal is marked with small circles, stretched over a punch, and deformation is observed in failure areas. FLD shows boundary between safe and failure zones.

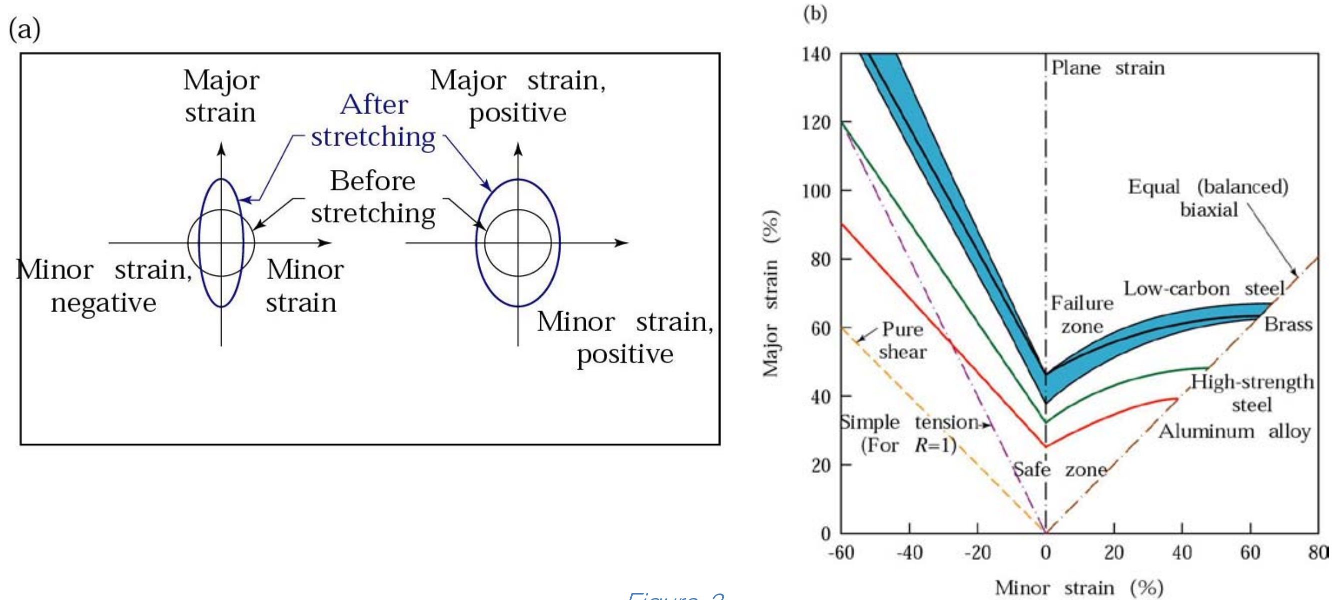


Figure 3

(a) Strains in deformed circular grid patterns. (b) Forming-limit diagrams (FLD) for various sheet metals. Although the major strain is always positive (stretching), the minor strain may be either positive or negative. In the lower left of the diagram, R is the normal anisotropy of the sheet.

III. Finite Element Metod

FEM is Method for numerical solution of field problems. In mathematics, finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems. It uses variational methods (the Calculus of variations) to minimize an error function and produce a stable solution. FEM cuts a structure into several elements (pieces of the structure). Then reconnects elements at "nodes" as if nodes were pins or drops of glue that hold elements together. This process results in a set of simultaneous algebraic equations. Method for numerical solution of field problems.

a) LS-DYNA

LS-DYNA is an advanced general-purpose multiphysics simulation software package developed by the Livermore Software Technology Corporation (LSTC). LS-DYNA is capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. LS-DYNA is optimized for shared and distributed memory UNIX, Linux, and Windows based, platforms. The code's origins lie in highly nonlinear, transient dynamic finite element analysis using explicit time integration.

LS-DYNA's potential applications are numerous and can be tailored to many fields. In a given simulation, any of LS-DYNA's many features can be combined to model a wide range of physical events. An example of a simulation, which involves a unique combination of features, is the NASA JPL Mars Pathfinder landing simulation which simulated the space probe's use of airbags to aid in its landing. LS-DYNA is one of the most flexible finite element analysis software packages available.

LS-DYNA consists of a single executable file and is entirely command line driven. Therefore all that is required to run LS-DYNA is a command shell, the executable, an input file, and enough free disk space to run the calculation. All input files are in simple ASCII format and thus can be prepared using any text editor. Input files can also be prepared with the aid of a graphical pre-processor.

There are many third party software products available for pre-processing LS-DYNA input files. LSTC also develops its own pre-processor, LS-Pre-Post, which is freely distributed and runs without a license. Licensees of LS-DYNA automatically have access to all of the program's capabilities, from simple linear static mechanical analysis up to advanced thermal and flow solving methods. Furthermore, they have full use of LS-OPT, a standalone design optimization and probabilistic analysis package with an interface to LS-DYNA.

One of LS-DYNA's most widely used applications is sheet metal forming. LS-DYNA accurately predicts the stresses and deformations experienced by the metal, and determine if the metal will fail. LS-DYNA supports adaptive remeshing and will refine the mesh during the analysis, as necessary, to increase accuracy and save time.

Metal forming applications include:

- Metal stamping
- Hydro forming
- Forging
- Deep drawing
- Multi-stage processes

Global Journal of Researches in Engineering (B) Volume XIII Issue I Version I Year 2013

IV. Problem Definition

The production of new stamping tools gets more troublesome, as the geometries get more complex and smaller tolerances on the dimensions are requested. This increases the risk for unexpected scenarios that could compromise the entire project. Thus, to deal with this, we wants to try a more advanced simulation tool, than the one-step solver they already have, to investigate what it could mean for future development of the company.

To investigate this, we want to use a simulation program in a real project to see its benefits at work and then evaluate if investment in a simulation tool and in personal would be profitable.

In this project the design of stamping tools, for stamping a girder, placed in a truck, will be considered, see figure (4). Note that the geometry is very complex and stamping in a single forming step is not possible but several stages will be necessary to form the part.

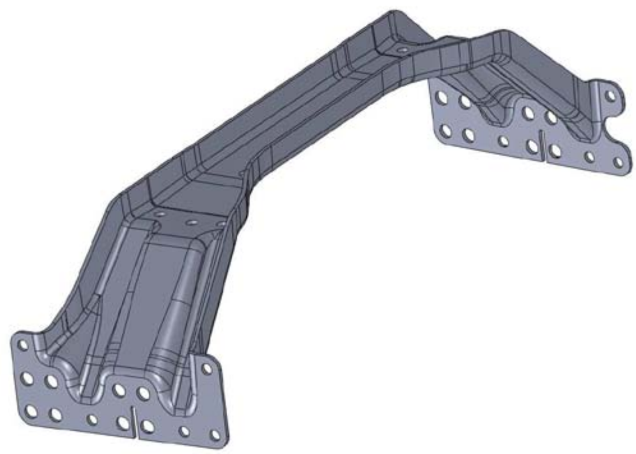


Figure 4 : Girder with sheet thickness 6 mm in Domex 500 MC

The girder has a very complex form and will, therefore, be formed in several stages.

V. Setting Up Simulation

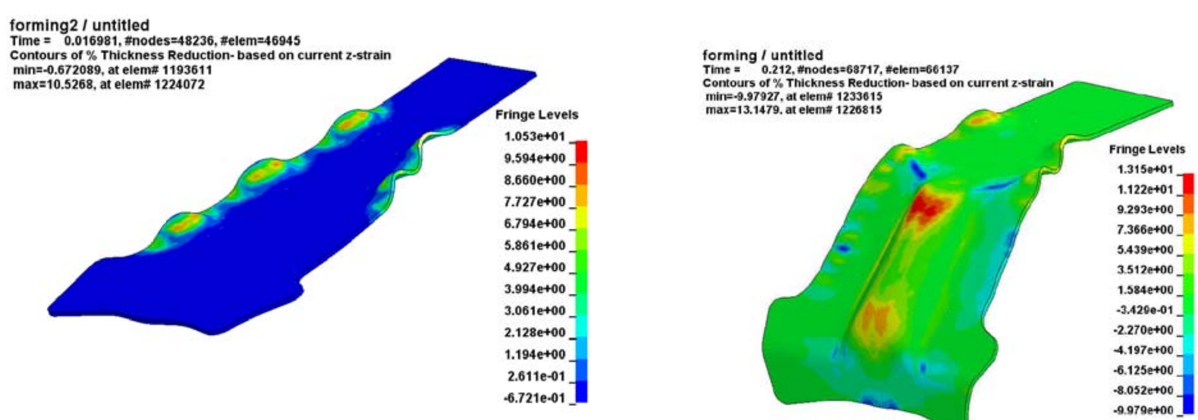
The steps for setting up a forming simulation in LS-DYNA are:

- Import the CAD-geometry into LS-DYNA and check surfaces.
- Create the blank or import it from previous forming step.
- Mesh the tool surfaces and the blank.
- Define material for the blank.
- Define the tool motions.
- Set up the controls for adaptive refinements, time-step etc.
- Run the simulations, for the trimming and forming steps, in LS-DYNA.
- Define constraints for spring back simulation in LS Pre-Post.
- Simulate the spring back.
- Evaluate the results in LS-Pre-Post.

This can further be generalised into the following three main steps: Pre-processing, Executing the simulation, Post processing.

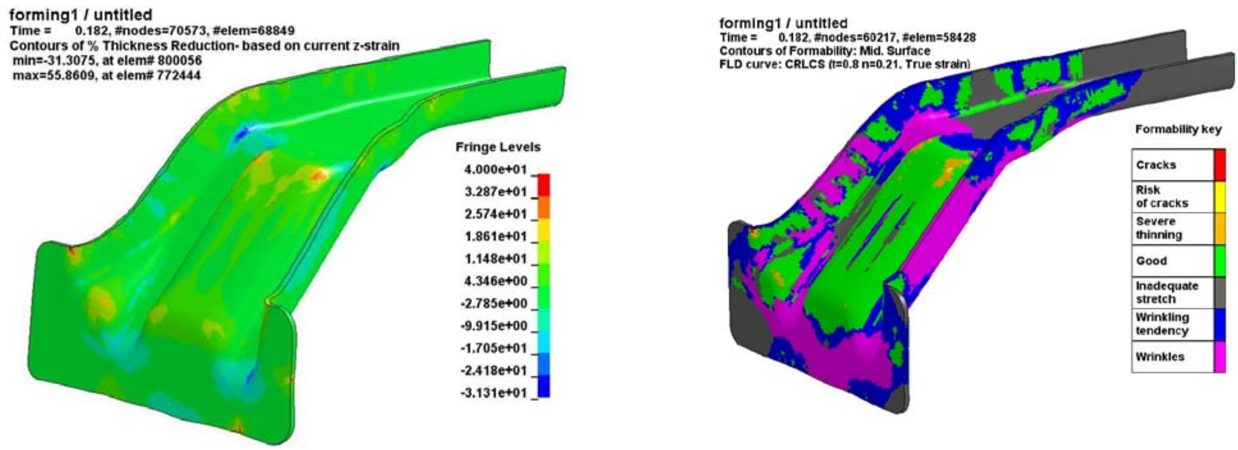
VI. Results

This section presents the final results for the forming simulations after spring back compensation.



(a) (b)

Global Journal of Researches in Engineering (B) Volume XIII Issue I Version I Year 2013
38



(c) (d)

Figure 5 : (a), (b): Thickness reduction in the first (pre-forming) step and the second forming step respectively. (c): Thickness reduction after the third forming step. The only region with cracks is at the small radii on the top of the flanges at the far end of the girder, but, as mentioned before, this was not considered since a design change will be made here. (d): FLD after the third forming step

To remove the crack on the right flange, in figure (5), an additional bump was drawn in the per-forming step. Figure (5) shows the results for the final forming simulations for all the forming steps. As can be seen in (c) and (d), the forming now looks acceptable and no cracks can be seen, except for the small radii on the top of the flanges at the far end of the girder. But, since a design change will be made here, this was left for now. What one could try to do is to draw additional bumps, in the pre-forming steps, here to avoid cracking.

After one has found a method where the forming is acceptable, i.e. does not yield cracks, too much wrinkling etc, one also has to look at the material distribution such that the formed part really has the shape requested and that the geometry is within the given tolerances. To do this, the model, after the simulations, was imported together with the CAD-model of the girder.The models were placed on top of each other to see where the formed part differed from CAD-model.The blank geometry was then slightly modified if required.

VII. Conclusions

There are some main conclusions that can be drawn from this project. Was the project successful? The main objectives were met and even though the hope was to be able to design the girder in two forming steps only, we finally reached a feasible solution. The main objective was to design the tools for forming the girder and then evaluate if investment in simulation software would be profitable. The answer to the question if this would be profitable, as can be understood from reading the discussion, depends greatly on how We chooses to profligate. If we focus on forming tools and especially advanced forming tools, an investment in simulation software would most certainly be profitable in terms of more orders, faster cheaper and safer production.

However, if we only produce a few forming tools per year, buying simulations externally, only when necessary, would maybe be a better option. But from a future perspective, one can expect the forming operations to be even more complicated and the need for simulation software will rise quickly as the demands on the products get higher.

Another important aspect that was learned during this project is that if one encounters problems during the forming with, for example, large cracks one usually has to make a drastic design change. Making small design changes usually have very small effect on the results One can try to alter the friction coefficient or try to explain the forming problems on some other small parameter, but the difference will be negligible. If there is a severe forming problem, something drastic has to be made.

Below are some of the main conclusions from this study listed:

- LS-DYNA has shown to be good for setting up forming simulations and also to simulate spring back.
- Designging complicated forming tools without simulation software compels the design engineer to work within in very narrow boundaries and only use ideas that have previously been tested.
- A simulation software is a must if the any company wants to be a leading company specialized on forming tools.

VIII. Acknowledgment

All praises and humble thanks are to Almighty ALLAH who gave me the valour and strength for successful completion of such a momentous work. No intellectual findings are the result of works of authors only; it includes also the precious contributions of others, which pave the path of success and immortality. Same is true for this research work.

My deep gratitude goes to my supervisor Professor Dr. TippaBhimasankara Rao for his initiation and supervision of this study and for his continuous support and engagement in the project. I believe that the effort embodies in this dissertation attained its present form only due to the vibrant and skillful leadership of him.

I would also like to express my gratitude to all those who have helped me with my work. Finally, and most importantly, I would like to express my warmest appreciation to my family and friends. Throughout this process, it has been a comforting feeling to have the support of loved ones that not only show genuine interest in my work, but who also help me to complete my work.

References Références Referencias

1. Livermore Software Technology Corporation (2006) Livermore Software Technology Corporation. *LS-DYNA Theory Manual*. Livermore Software Technology Corporation, March 2006.
2. Metal Forming Analysis BY R. H. Wagoner, J.-L. Chenot Cambridge University Press, 07-May-2001.
3. Fundamentals of Metal Forming Processes by B L Juneja Publisher, New Age International, 2006.
4. Sheet Metal Forming Analysis Using FEM: Prediction of Spring Back in Edge Bending Process Using FEM By AlieWubeDametew, DametewAlieWube Lambert Academic Publishing, 2012.
5. Sheet Metal Forming Processes Die Design By VukotaBoljanovic Industrial Press Inc., 2004
6. Lee, D.W. & Yang, D.Y. Consideration of geometric nonlinearity in rigid-plastic finite element formulation of continuum elements for large deformation.Inc. J. Mech. Sci., 1997, 39, 1423-440.
7. Murray, D. W. & Wilson, E.L. Finite element large deflection analysis of plates. J. Engg. Mech. Div. Proc., 1969. ASCE 95, 143-65.
8. ASTM. ASTM E2218-02, JAN 2012. URL http://www.astm.org/ Standards/E2218.htm.
9. Larson and Bengzon (2010) Mats G. Larson and Fredrik Bengzon. The Finite Element Method: Theory, Implementation and Practice. Springer, January 2010.
10. Roll,K.,Faust, A. Forming simulation of highstren-gth steels with direction specific hardening Mode-lling.Proceedings,Auto Met Form, Wroclaw, May 2008.

Global Journals Inc. (US) Guidelines Handbook 2013

www.GlobalJournals.org