Finite Element Modeling For Simulation Of A Rectangular Tank With An Integral Flange For Automotive Application.

Ranganath J[1], Rajashekhar Hosalli[2], C. P. S. Prakash[3]
[1]*M.Tech (CIM) Student, Department of Mechanical Engineering, Dayananda Sagar College of Engineering, Bangalore-560078, India*
[2]*Senior Lecturer, Department of Mechanical Engineering, Dayananda Sagar College of Engineering, Bangalore-560078, India*
[3]*Professor & Head, Department of Mechanical Engineering, Dayananda Sagar College of Engineering, Bangalore-560078, India*

Abstract

Minimization of response times and costs and maximization of the efficiency and quality in producing a product are imperative for survival in the competitive manufacturing industry. Sheet metal forming is a widely used and costly manufacturing process, to which these considerations apply. Weight reduction while maintaining functionality is one of major engineering goals in product design. Magnesium sheet becomes favorable compared to steel, regards to some advantages like less density, free from casting defects etc. This work will be done to study the finite element (elastic-plastic) analysis of sheet metal forming process using the finite element software. Ansys simulation is carried out to gain accurate and critical understanding of sheet forming process.

Keywords: sheet metal forming, FE simulation, deep drawing process

1. Introduction

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.

In the manufacturing process based on numerical standards, simulation is an important tool in applying the experience and knowledge gained in the past to the development of new models. A limiting factor in design process is the necessity of producing the desired shape in a single deep drawing for example, with no cracks or wrinkles in the sheet metal. As automotive and aerospace industries are growing rapidly the demand for precise and accurate information concerning parts design and formability of metal sheet becomes essential. Sheet metal forming simulation plays an indispensable role in integrating manufacturing necessities into the product design process at an early stage. In conjunction with concurrent engineering, sheet metal forming simulation is proving to be an important tool in linking design and manufacturing. Since sheet metals are most susceptible to failure under plane strain condition, several plane strain membrane finite element codes for sheet metal forming analysis has been developed by M. J. Saran and Wagoner, W. H. Frey and Wenner, M. P. Sklad . These codes, in the absence of shape tooling curvatures, are capable of predicting deformation strains very accurately, but as tooling curvature becomes larger to cause significant bending strains, these codes can no longer be capable to predict accurately. The objects of this works will focus on simulating nonlinear sheet forming process using finite element technique in order to come up with a clear and better understanding of metal flow of sheet metal forming process.

In the present work, sheet metal is formed to a rectangular tank of 220 mm length by 110 mm width. The deformation is for 15.6mm. Due to symmetry, a quarter geometry of the punch, container and sheet metal are built using Ansys mixed approach. The objects are meshed with solid185 elements. The interface of punch and sheet metal, sheet metal and containers are defined with contact elements. The displacement load is applied to obtain faster convergence. The simulation is carried out for warm sheet metal formation under four temperature related material stress-strain data. The results for vonmises stresses, displacements, contact pressure and loads are obtained for same depth of sheet metal formation. The results shows that the temperature plays important role in reducing load requirements in forming to the required shape due to its lower yield point and higher strain with lesser stresses. Also contact pressure values also reduce with increase in temperature. This helps in reduction of die strength requirements. Therefore lower cross section of dies can be used. Even the stress reduction with higher strains helps to get reduced load requirements of punch. So finite element simulation helps to estimate the load requirements along with simulation to estimate possible problems before the actual model in production.

2. Problem Definition and Finite Element Model Development.

2.1 Problem Definition

Magnesium Sheet metal formation with warm deep drawing process for different thermal material data with uniform LDR ratio is the main definition of the problem. Critical region identification in the deep drawing process is also another definition of the problem. Magnesium is having low density and magnesium products are better than steel products due to its defect free sheet metal formation unlike steel casting. But forming is a difficult process for magnesium at room temperatures. But at higher temperature, the magnesium can be easily formed to the required shape. 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.

The project objectives include:

- Deep drawing simulation through Finite element software
- Estimation of thermal effects on structural deformation and resultant stresses
- Identification of critical regions of possible cracks
- Identification of load requirements
- Effect of Deformation on stress generation.

2.2 Methodology

- Initial built up of geometry of movable die, fixed die and sheet metal

ISSN: 2278-0181
Vol. 1 Issue 5, July - 2012

- Meshing with three dimensional elements
- Contact pair creation between fixed die, movable die and sheet metal
- Application of displacement field based on the type of analysis
- Application of nonlinear properties
- Solving the problem with different temperature dependent material data.
- Analyzing the problem
- Results presentation

2.3 Load data & Material Properties

Rectangular Punch : 220X110mm

Punch corner radius=12mm

Blank Material=AZ31B

Thickness =1mm

Young's Modulus=44.8Gpa

Poison's ratio=0.35

Friction coefficient =0.1

Thermal Properties:

Thermal conductivity =159 w/s ^{0}c

Heat capacity=1.7676w/mm^{2} c

Thermal conductivity of tool =60.5w/s ^{0}c.

Heat capacity of tool=3.41w/mm^{2} c

Convection coefficient =0.03w/mm^{2} ^{0}c.

Interface transfer coefficient =4.5 w/mm^{2} ^{0}c.

Factor to convert plastic deformation energy to heat=0.95

2.4 Load Cases For The Problem

Case	LDR	Die, Blank and holder temperature
A1	2.3	150
A2	2.3	200
A3	2.3	250
A4	2.3	300

Table 2.1 : Load Cases

The above load cases are considered to find the punch load requirements, contact pressure etc. The temperature dependent properties are considered for analysi

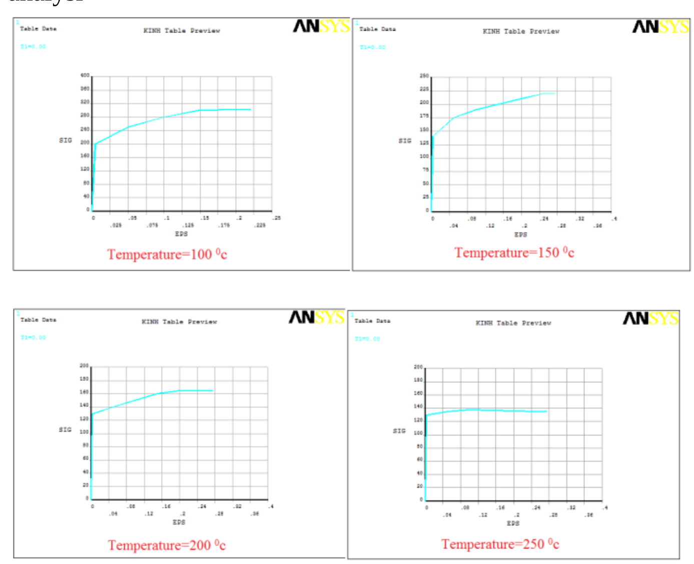

Fig 2.1: Material Data curves

The figure 2.1 shows stress strain data for different temperature loads. The yield stress is reducing with increased temperature along with reduction of slope. The graphs shows almost a flat curve for high temperature of 250 ^{0}. This slope which generally represented by plastic modulus is a stain hardening parameter which gives resistance for deformation.

2.5 Geometry Of The Problem

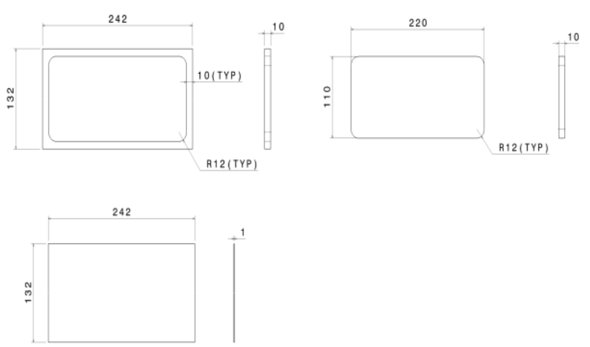

Fig 2.2: Dimensional view of the Punch, Blank Holder and Sheet Metal

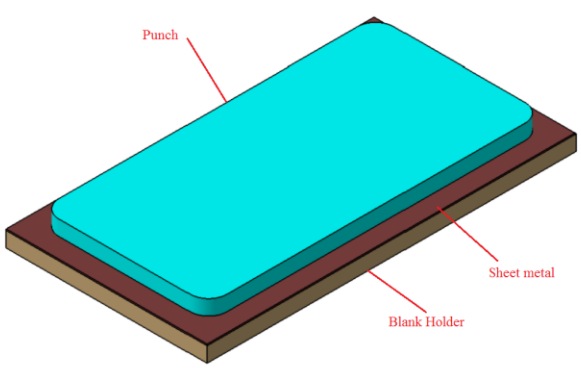

Fig 2.3: Geometrical view of the problem

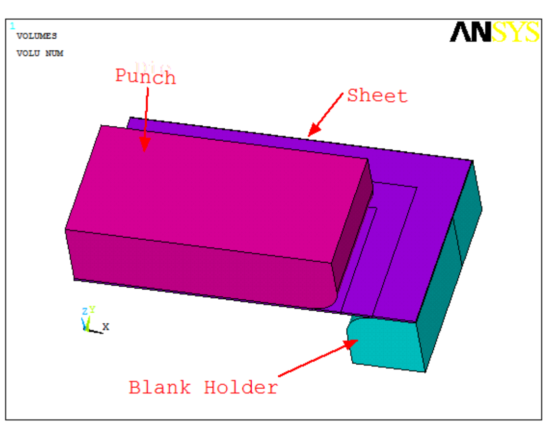

Fig 2.4 : Geometrical built up

The figure 2.2 & 2.3 shows geometry built up of the die. The dimensions are shown in the picture. All dimensions are represented in mm. The curvature is provided as per the requirements. Sheet metal in rectangular form is represented with 1mm thickness. The thin strip of low strength material will be shaped as per the die shape. The catia software is used to built the geometry for clear understanding of the three dimensional problem.

The figure 2.4 shows the geometrical built up of the problem. Ansys mixed up approach is used to built the geometry. Boolean operations are used to built the curved geometries. The structure is divided to ease map meshing of the problem. Workplane options are used to divide the structure. Quarter geometries with fillet regions for punch, sheet metal and blank holder are shown in the picture.

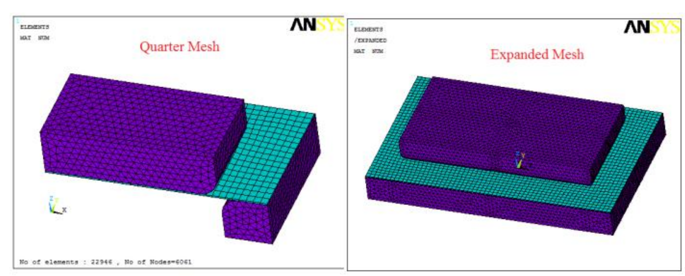

Fig 2.5 : Meshed plot of the problem

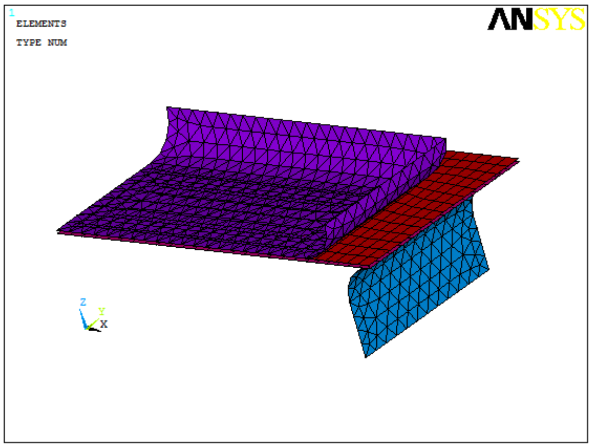

Fig 2.6 : Contact definitions

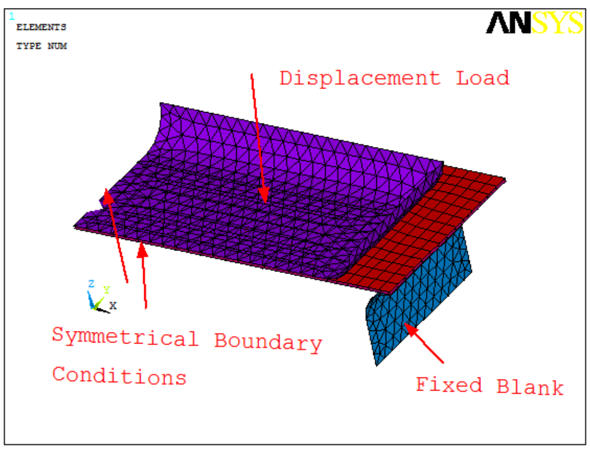

Fig 2.7 : Boundary conditions of the problem

The figure 2.5 shows meshed model of the problem. A 4 noded tetrahedral (Solid185) is used to mesh the structure. The structure is tetra meshed with high density at the curved regions. Both steel dies and magnesium sheet are represented with different colors. Generally map mesh is good for accurate results but high density of mesh is not possible with map mesh. Both die and sheet metal are meshed with different materials. The colors shown in the figures represents this change in mesh.

The figure 2.6 shows contacts elements created in the nonlinear problem. Top punch bottom surface and top surface of the sheet metal are created with one contact pair and the bottom area of sheet metal with top surface of bottom die are created with another pair. Ansys contact manager is an useful tool for building this contact pairs which are difficult with earlier versions. Contact friction also can specify through the contact manager. Targe170 and Contac174 elements are automatically built for the pairs for contact estimation. The penetration tolerance is reduced for better convergence. Augmented lagrangian algorithm with standard contact options are used for convergence. Also in the nonlinear options, displacement convergence is specified for faster execution. Generally contact problems based on both geometrical and material nonlinearity consumes heavy resources of computer space, time along with convergence problems.

The figure 2.7 shows boundary conditions of the problem. Since quarter geometry is built symmetrical boundary conditions are applied at the edges. The bottom blank holder is completely fixed in position. The punch nodes are applied with displacement loads. To save solution time with faster convergence, the punch and blank holder mesh is cleared after contact pairs are created. This helps in obtaining target and contact elements. Since both members (punch and blank holder) are considered rigid, the clearing of elements helps in faster iteration of the problem. So more steps can be considered for better solution. So the displacement load is applied for the top target elements which are created from the contacting surface of the punch. Target elements of blank holder are fixed in all degree of freedom.

2.6 Assumptions

- Materials are assumed homogenous and isotropic up to yield point
- Nonlinear domain is applied after yield stress

ISSN: 2278-0181
Vol. 1 Issue 5, July - 2012

- Contact elements are defined to obtain the solution, which employ iterative solvers for execution
- All Finite element approximations are applied during solving
- Multi-linearity is assumed for nonlinear stress strain curve.

3. Results & Discussion

The rectangular tank sheet formation is carried out in number of iterations. The formation of sheet metal along with resulting stresses are represented as shown in the following figures.

Case 1 : Results Analysis(Case 1 – 100 ^{0}c)

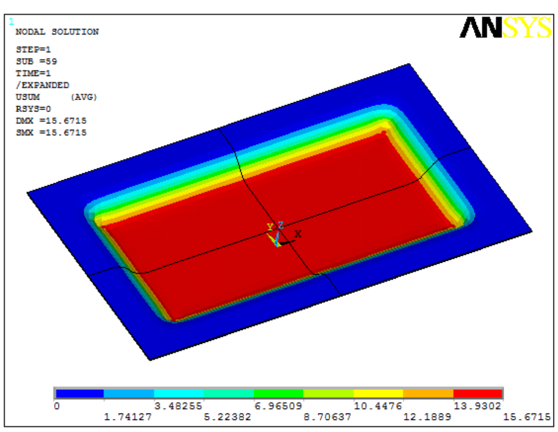

Fig 3.1 : Displacement plot

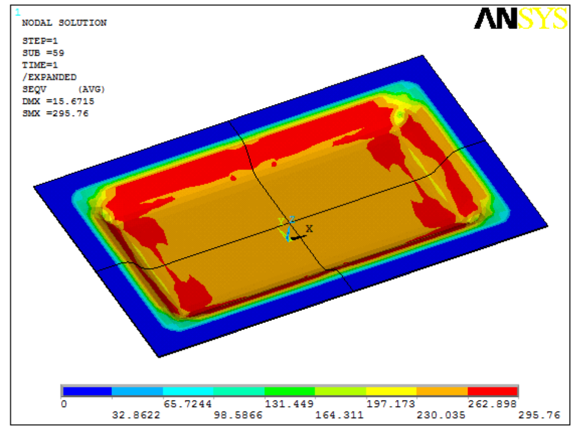

Fig 3.2: Vonmises Stress in the problem

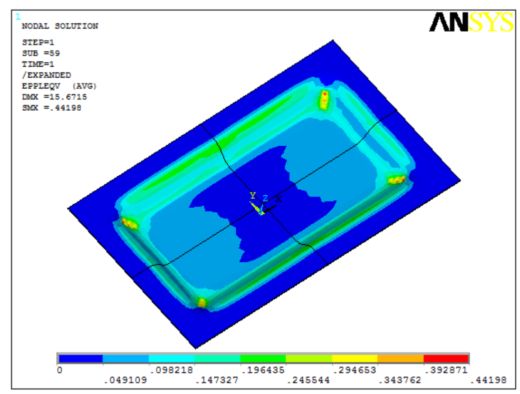

Fig 3.3: Plastic Strain

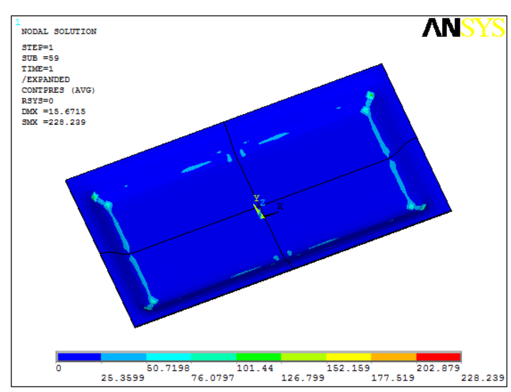

Fig 3.4: Contact Pressure

The figure 3.1 shows deformation of 15.6715 mm due to punch movement. Maximum deformation is observed at the bottom and minimum deflections at the top. The status bar at the base shows variation of displacements. A gradual change of deformation can be observed. The displacement convergence used in the problem helps in obtaining the deformation in the sheet metal. Totally 59 iterations are carried out to obtain the required deformation of 15.6mm. In each step, a small incremental displacement is applied on the sheet metal.

The figure 3.2 shows the developed vonmises stress in the deep drawing process. Maximum vonmises stress of 295.764 Mpa can be observed at the end of sheet metal formation. The red colour region shows maximum stress region. The blue colour region shows minimum stresses in the structure. Vonmises stress is the limiting stresses for plastic flow. The stress increase is decided by the material nonlinear nature after yielding. Nonlinear curve nature is decided by the strain effect on the material. Higher slope in the nonlinear regions increases the stress and flat slope reduces the stresses in the member.

The figure 3.3 shows plastic strain in the problem due to deep drawing process. Mainly plastic strains are concentrated around the bottom corners. This may be due to sharp variation of geometry by which flow lines are converging. Totally a maximum strain of 0.44198 can be observed which is less then the allowable critical plastic strain of 2. So deep drawing process is safe.
The presence of plastic strain represents the yielding of the structure.

The figure 3.4 shows contact pressure developed in the structure. Maximum contact pressure is around 228.239Mpa at the corners. Maximum contact pressures are observed at the bottom corners. The contact pressure is displaced by plotting only contac174 elements and hiding other elements from display. Higher contact pressure indicates the regions to modify to avoid stress concentration. Contact pressure directly affects the die design. A rigid die requirement is more with higher contact pressure to avoid deflection in the die which miserably affects the surface accuracy of the formed object.

Case 2:Results Analysis(Case 4 – 250 ^{0}c) [Final]

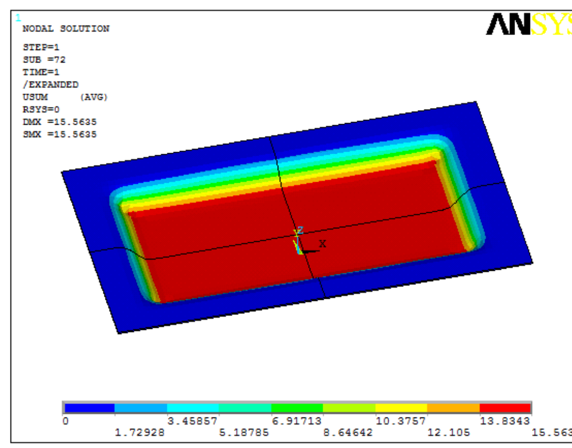

Fig 3.5 : Final Deformation plot

The figure 3.5 shows deformation of 15.5635 mm due to punch movement. Maximum deformation is observed at the bottom and minimum deflections at the top. The status bar at the base shows variation of displacements. A gradual change of deformation can be observed. Totally 72 iterations are carried out to obtain the required deformation of 15.5635mm. In each step, a small incremental displacement is applied on the sheet metal.

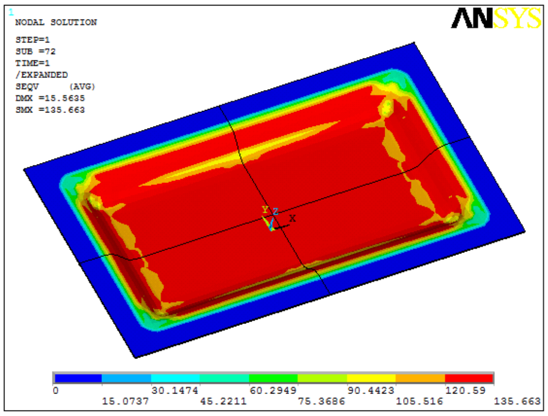

Fig 3.6: Vonmises Stress in the problem

The figure 3.6 shows the developed stress in the deep drawing process. Maximum vonmises stress of 135.663 Mpa can be observed at the end of sheet metal formation. The red colour region shows maximum stress region. The blue colour region shows minimum stresses in the structure. The drop of stress can be observed due to the reduction in yield stress and other stress formation with strain under raise of temperature.

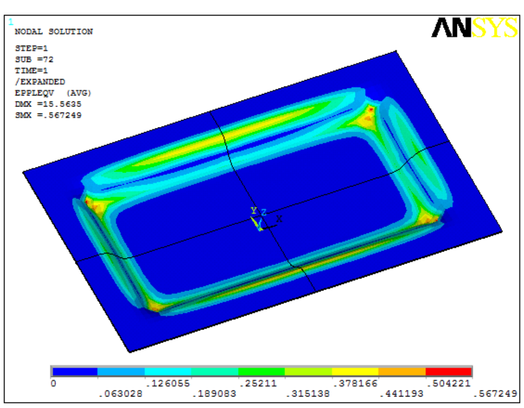

Fig 3.7: Plastic Strain

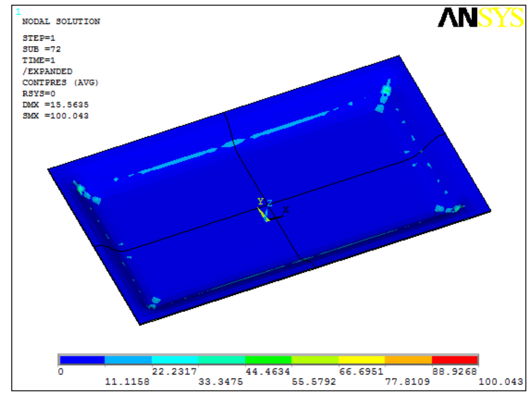

Fig 3.8: Contact Pressure

The figure 3.7 shows plastic strain in the problem due to deep drawing process. Mainly plastic strains are concentrated around the bottom corners. Totally a maximum strain of 0.567249 can be observed which is less then the allowable critical plastic strain of 2. Higher plastic strain regions should be carefully treated to prevent any possible crack formation and residual stresses.

The figure 3.8 shows contact pressure developed in the structure. Maximum contact pressure is around 100.043 Mpa at the corners. Maximum contact pressures are observed at the bottom corners. The contact pressure is displaced by plotting only contac174 elements and hiding other elements from display. From the observation the contact pressure is mainly limited to bottom fillet regions.

4. Conclusions & Future Scope

4.1 Conclusions

The magnesium rectangular pan formation is analysed using finite element software Ansys. The results summary is as follows.

- Initially the punch, magnesium sheet and fixed die or blank holder are modeled as per the specifications. Later the structure is meshed with solid185 tetra hedra elements to get dense mesh at the curvature regions. Contact pairs are created between punch, sheet metal interface, die , sheet metal interface using Targe170 and Contac174 elements. The displacement load is applied and problem is executed in the nonlinear domain using material properties specified for given temperature range.

- Analysis has been carried out for load requirements for sheet metal formation. The results shows reduced load requirements with temperature effects. The temperature reduces the yield stress along with stress reduction with given plastic strains by which resisting load also is reducing. The stress values for vonmises and contact pressure are also reducing with increased temperature. From the finite element simulation, the region of thinning and probable regions of failure regions can be identified. Higher stress and plastic strain regions are the major regions of failures. Finite element simulation helps in avoiding prototype built up and checking for the required load calculations.

- Finite element software is effectively implemented for sheet metal formation which saves the time of prototype testing to find the required loads and stress conditions. Also finite element solution helps to find stress,strain deformation condition at each stage along with all critical regions which is not possible with practical testing. So a better product can be built by effectively utilizing finite element analysis. All the necessary results for analysis are represented to find structural conditions during forming process.

4.2 Future Scope

- Ansys-Ls Dyna solvers can be used to improve further the efficiency of the system
- Residual stress estimations can be carried out
- Sheet metal thickness requirements can be carried out for the required shape
- Optimisation of dies can be carried out using the computer simulation.
- Transient response in coupled field domain can be carried out.
- Flow simulation at higher temperatures using Fluent software can be carried out.

REFERENCES

[1] Hariharasudhan Palaniswamy, Gracious Ngaile, Taylan Altan, Finite element simulation of magnesium alloy sheet forming at elevated temperatures, Journal of Materials Processing Technology 146 (2004) 52-60 .

[2] Hakim S. Sultan Aljibori," : " Finite element Analysis of Sheet Metal Forming Process", European Journal of Scientific Research, ISSN 1450-216X Vol.33 No.1 (2009), pp.57-69.

[3] Y. Park, J. S. Colton, "Failure Analysis Of Rapid Prototyped Tooling In Sheet Metal Forming—V-Die Bending", Journal Of Manufacturing Science And Engineering, February 2005, Vol. 127 [4] Fahd Fathi Ahmed Abd El Au,"Finite Element And Experimental Studies Of Springback In Sheet Metal Forming", National Library Of Canada,2003

[5] Alejandro Quesada,"Influence Of The Parameters Of The Material Model In Finite Element Simulation

Of Sheet Metal Stamping", Euro-mech Solid Mechanics Conference J. Ambrósio Et.Al. (Eds.), Lisbon, Portugal, September 7-11, 2009

[6] SRIPATI SAH, " Investigation Of Contact Pressure Distribution On Sheet Metal Stamping Tooling Interfaces: Surface Modeling, Simulations, And Experiments" ,University of Massachusetts,2007",

[7] I. Pahole, " Bending of sheet metal of complicated shapes (for 90o angle and more) in combined tools" ,.Journal of Achievements in Materials,and Manufacturing Engineering, VOLUME 16, ISSUE 1-2, May-June 2006.