A Finite Elemental Study of Contact Pressure Distribution in Stamping Operations

Shambhuraje Jagatap [1,*], Prof. Bharat S. Kodli [2]
[1*]PG Student, production Engineering, PDA College of Engineering, Gulbarga585102, Karnataka (INDIA)
[2]Professor, department of mechanical Engineering, PDA College of Engineering, Gulbarga585102
Karnataka (INDIA)

Abstract

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 analysis, 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. Contac174 and Targe170 elements are defined for contact definitions. The problem is converged at different steps and results are captured for stress and plastic strain effects. Final punch load comparison is done between experimental and numerical solutions. The results show slight variation of numerical and finite element results. Further two dimensional analysis for stamping also shows higher number of steps for stamping operation. The results are captured for vonmises, displacements, plastic stain 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.

Keywords Sheet metal stamping, contact pressure distribution, pressure surface

1. Introduction

The work was carried out to study the finite element (elastic-plastic) analysis of sheet metal forming process using the finite element software. Axisymmetry element mesh and plain strain element mesh were use incorporated with slideline features to model and study the sheet metal forming process. Simulation of elasticplastic behavior of magnisium sheet was carried out under non-linear condition to investigate sheet metal forming process. 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. Magnisium sheet becomes favourable compare to steel regards to some improvement at aerodynamic designs, increased engine efficiency and fuel economy. Wide range of magnisium automotive product included doors, fenders, bumpers face bars, seat frames and roof panels have been produced.

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 analyzing the contact pressure distribution directly at the tooling-workpiece interface.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 know-how 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 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.

2. Methodology

Here two ways of stamping process is 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. Initial built up of geometry of movable die, fixed die and sheet metal

- 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

3. Modeling of pressure distribution

3.1 Geometric model (3D)

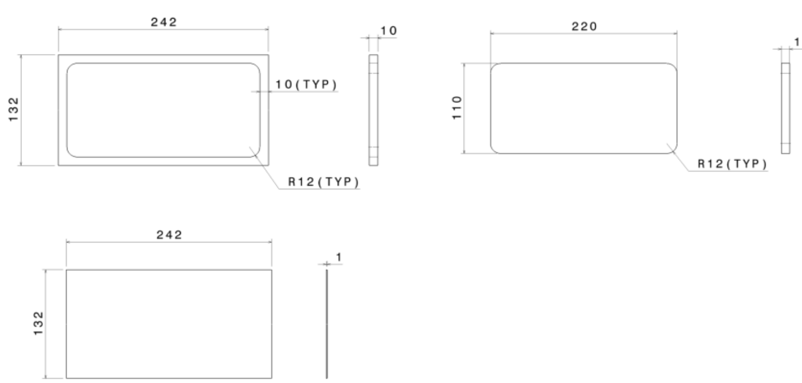

Figure 1.

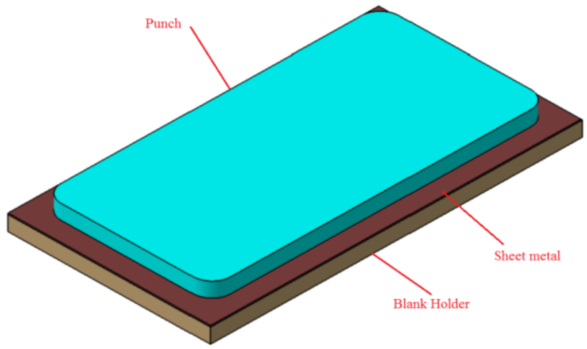

Figure 2.

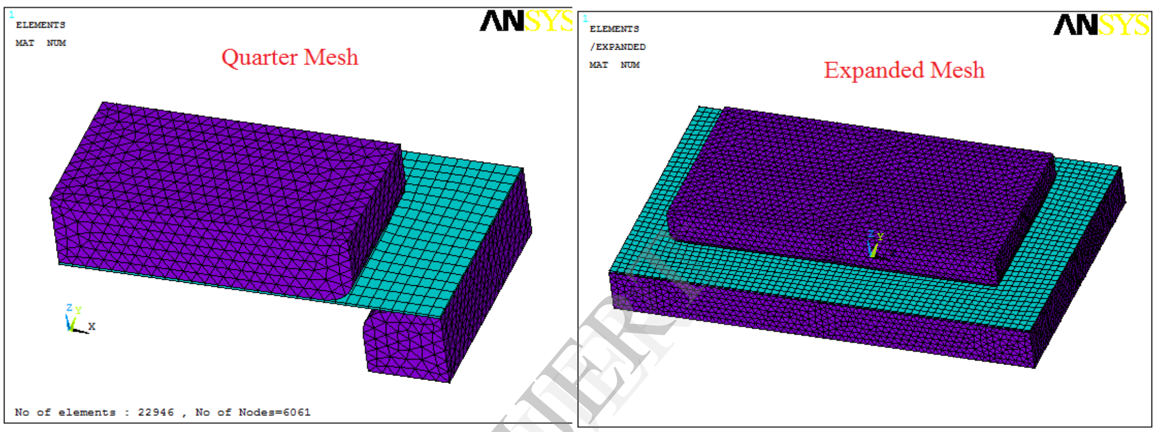

Figure 3.

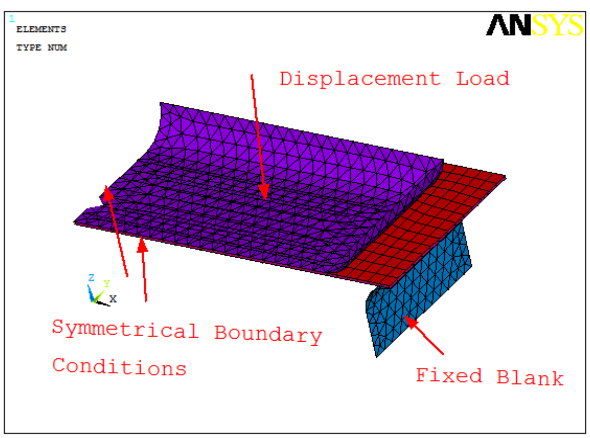

Figure 4.

The figure2 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 figure3 shows geometrical built 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.

The figure4 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 represent this change in mesh.

The figure 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 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.

3.2 Geometric model (2D)

Stamping simulation is carried out by another two dimensional model. The modeled geometry using cad modeling software is as follows.

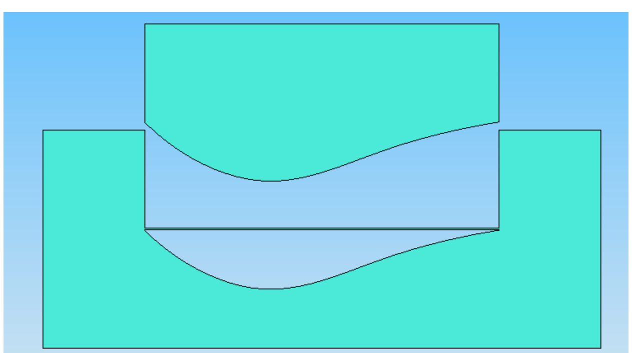

Figure 5.

The geometry is built using points and spline option. Rectangular geometry is directly built and other parts are built using spline and line options.

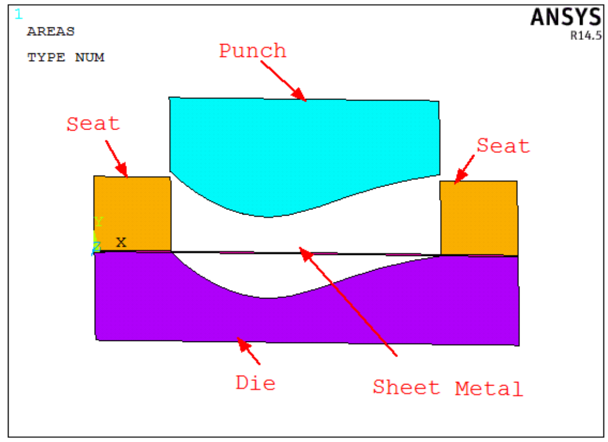

Figure 6.

The above figure shows identification of different parts in the simulation process. The major parts are die, sheet metal, punch and seat. Here punch is fixed in position and punch moves in the required direction. The geometry is represented by different colours to identify the parts.

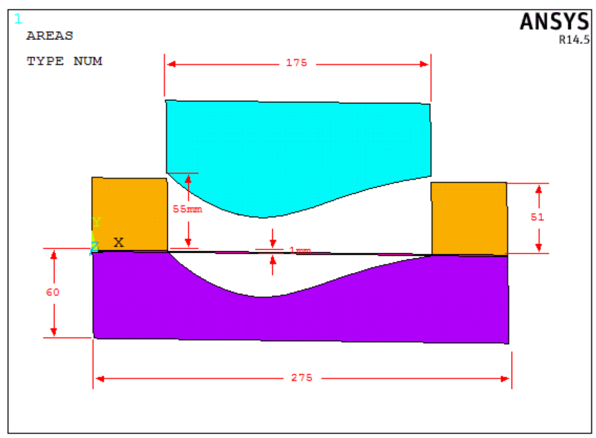

Figure 7.

The figure 7. Shows dimensional plot of the problem. Maximum dimension of the die is 275mm and height of the die is 60mm. The punch is going to move for 55mm to form the sheet metal to the required shape during stamping operation. The geometry is split to ease map meshing of the geometry. Generally map mesh gives better results compared to the free mesh due to regular nature of elements. This regular element satisfies the requirement of aspect ratio, warpage, skewangle, minimum and maximum angle quad. Also

jacobian is checked for the quality of the mesh. Plane 182 element properties are attached with plane stress with a thickness of 25 mm for the sheet metal.
Two sets of contact pairs are defined between the punch and sheet metal and sheet metal and die surface. Targe169 elements are used for rigid elements and contac172 is defined for the contact region. The normals are shown in the figure. These contact elements use augmented lagrangian algorithm for proper convergence.

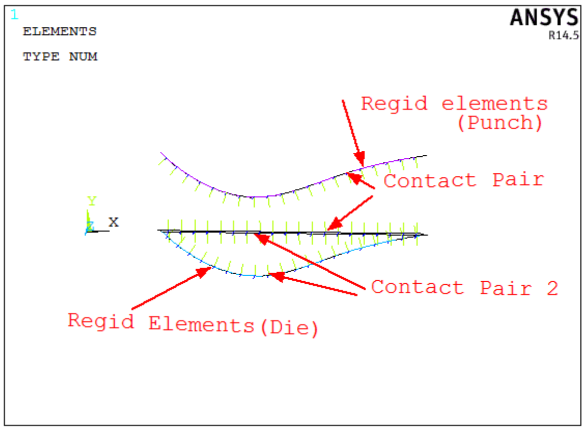

Figure 8.

3.3. Assumptions

- Materials are assumed homogenous and isotropic up to yieldpoint.
- Nonlinear domain is applied after yield stress.
- 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 curve.
- Both two dimensional and three dimensional contact elements are defined for the problem. Conac172, contac174, Targe169 and Targe170 elements are used for the problem.

4. Finite element model development

3.1 Load Cases for the Three dimensional problem:

Case	Die, Blank and holder temperature
A1	100
A2	150
A3	200
A4	250

Table1: Load Cases

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

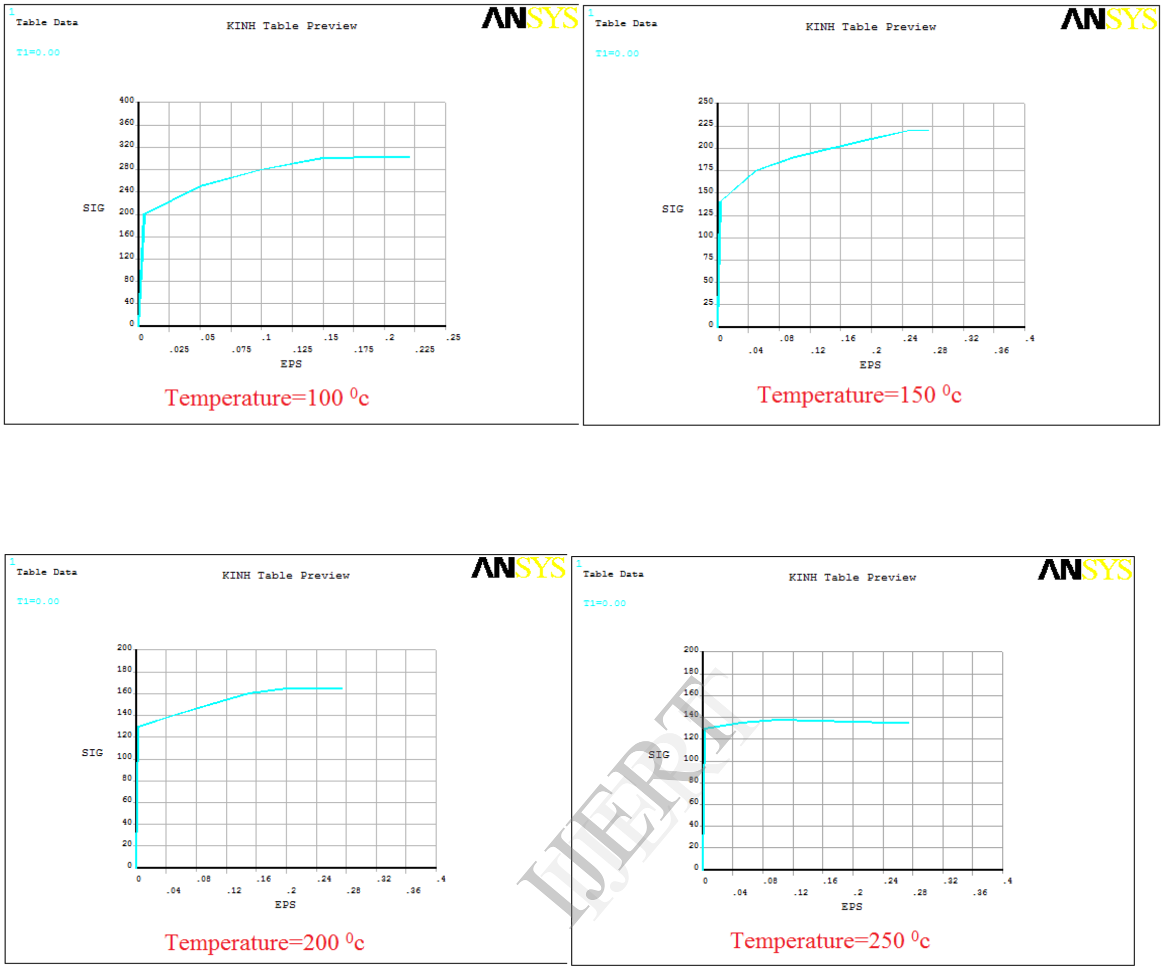

Fig 9. Material Data curves

The above figure 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.

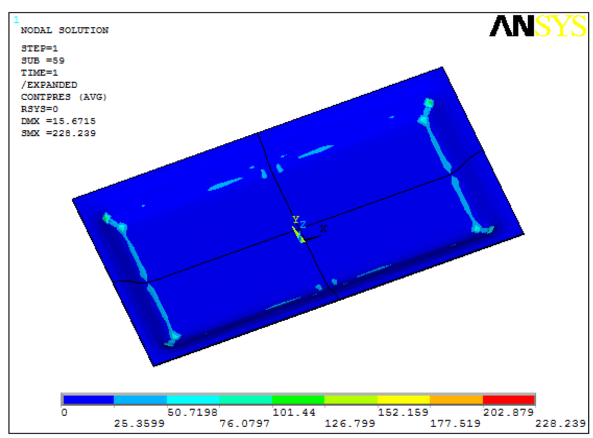

Figure 10.

5. Results

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

The figure 10. 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 2 – 150 ^{0}c)

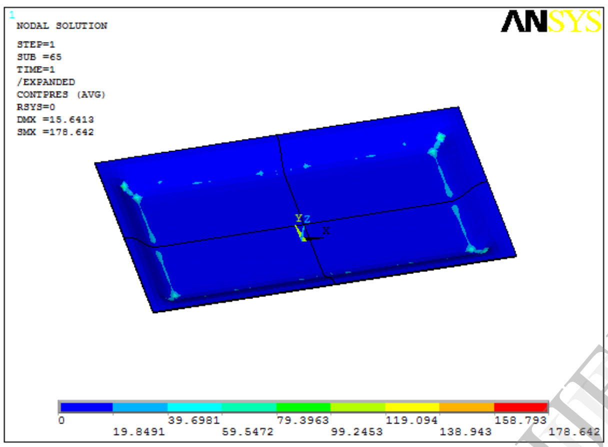

Figure 11.

The figure shows contact pressure developed in the structure. Maximum contact pressure is around 178.642Mpa 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. The reduction in contact pressure is attributed to reduced yield stress of the structure and stress strain slope in the plastic region.

Case 3. Results Analysis (Case 3 – 200 ^{0}c)

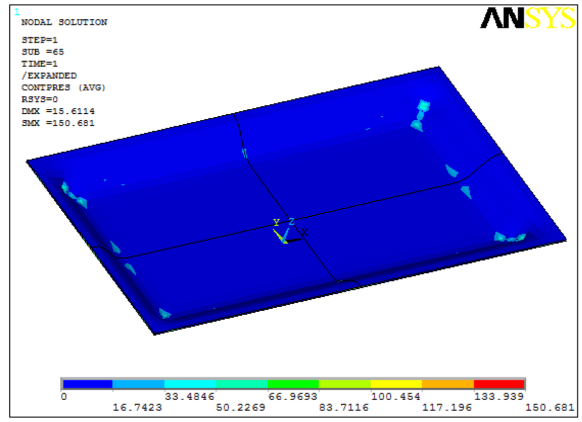

Figure 12.

The figure 12. Shows contact pressure developed in the structure. Maximum contact pressure is around 150.681 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. The reduction in contact pressure is attributed to reduced yield and stress values for the given strain with increased temperature.

Case 4. Results Analysis (Case 4 – 250 ^{0}c)

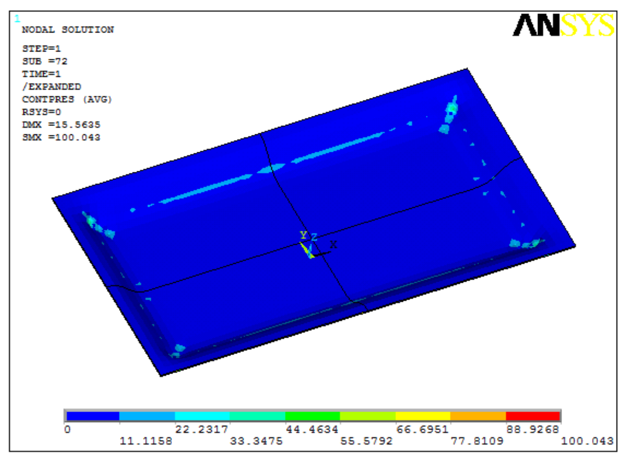

Figure 13.

The figure 13. 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.

Temperature	Vonmises Stress(Mpa)	Plastic Strain	Contact Pressure(Mpa)
100	295	0.44198	228.239
150	203.8	0.43067	178.6
200	162.18	0.47132	150.68
250	135.66	0.5672	100.143

Table 2.

The table 2. Shows reduced vonmises, contact pressure with the increase in the temperatures. The result shows reduction in resistance of the metal flow under temperature effects.

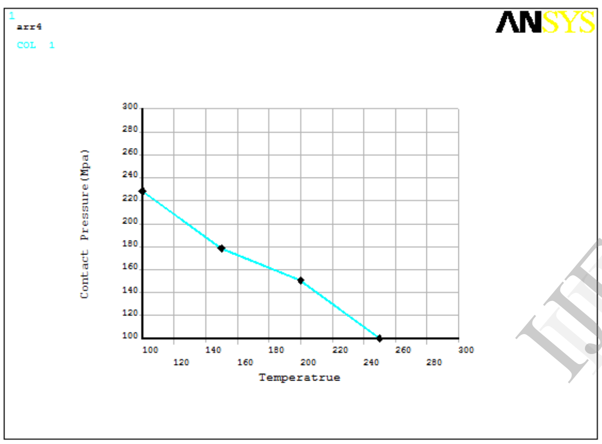

Fig 14: Contact pressure variation with reference to Temperature

The figure 14. Shows reduction in contact pressure with the increase in temperature load. This helps to reduce the burden of design of high rigid dies. So cost also will reduce by saving the die material.

Case 5. 2 dimensional analysis with steel plate (thickness of the plate: 1mm)

Further analysis has been carried out with two dimensional domain to find the contact nature with irregular systems. So an irregular geometry is built and nonlinear contact simulation through implicit algorithm is done. The simulation results are as follows

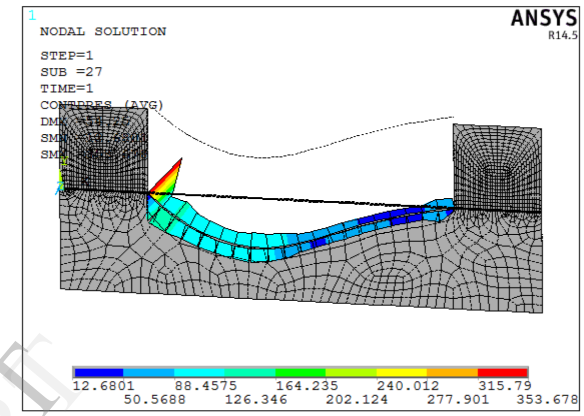

Figure 15.

The figure4.37 shows contact pressure development in the problem. Maximum contact pressure 353.678Mpa. Maximum contact pressure is taking place at the left side corner due to irregular shape of the formed object. Lesser contact pressure is observed at the left end and as shown blue colour. Different colors shows variation of contact pressure across the geometry.

Final deformation in the structure. Maximum deformation is shown as 54.25mm. The figure shows complete formation on the die surface can be observed. Maximum deflection is shown by red color. So irregular objects also can be simulated through finite element simulations.

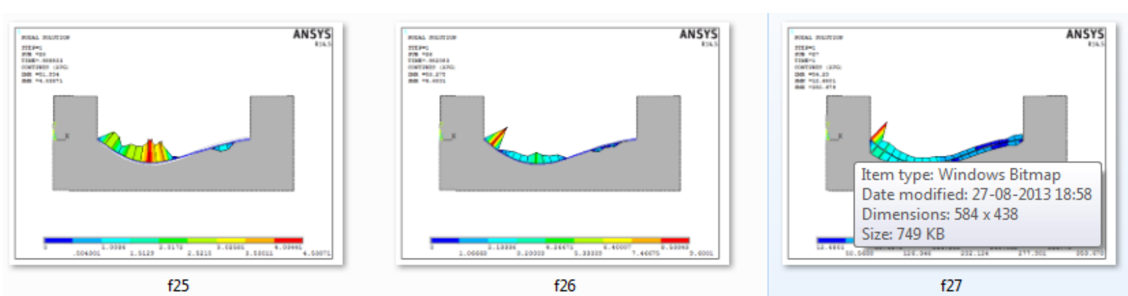

Figure 16.

The figure 16. Shows contact pressure simulation in the process after contact is initiated. A gradual increase of contact pressure can be observed in the problem. Spread of contact pressure also observed from the contact simulation. Complete spread of contact pressure can be observed at the end.

6. Conclusions:

Simulation for stamping process is carried out and checked for Two dimensional and three dimensional the deformation, stress, plastic strain and contact pressure. The overall process is as follows. Initially the punch, magnesium sheet and fixed die or blank holder is modeled as per the specifications. Later the structure is meshed with solid185 tetrahedra 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 result 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 are 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.

A two dimensional stamping simulation is carried out with plane stress approach after building the geometry in the cad modeling software.

Contact definition is carried out between punch, sheet metal and die, sheet metal interfaces. Contac172 and Targe169 elements are defined with reduced penalty stiffness values for better convergence.

7. References

[1] 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.

[2] 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

[3] Fahd Fathi Ahmed Abd El Au,"Finite Element And Experimental Studies Of Springback In Sheet Metal Forming", National Library Of Canada,2003

[4] 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

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

[6] 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