IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
ISSN: 2278-1684 Volume 4, Issue 5 (Nov. - Dec. 2012), PP 15-22
www.iosrjournals.org

Determination of Strain Distribution in Single Stage Deep Drawing Process by Finite Element Simulation

Shishir Anwekar[1], Abhishek Jain[2]
[1, 2] Mechanical Engineering, University Institute of Technology, Barkatullah University, Bhopal,India

Abstract: *Modern production is based on the increased analysis of forming processes in digital environment before the actual production set-up to avoid the costly and complex experiments. The main aim of presented work is to approach the single stage deep drawing process of thin walled, mild steel, conical back plate of radial impeller of blowers by means of a finite element analysis. In the presented study, simulation of the drawing process for determining strain distribution pattern in the drawn component for a particular displacement is explained. The study was conducted by using ANSYS12.0, in which, two models have been tested. Both models constructed solely out of axisymmetric, quad 4 node, PLANE 42 elements. These axisymmetric models have been used to simulate the drawing process of drawing quality mild steel IS2062 grade. The experimental analysis is carried out on two different flat plates having thickness 3 mm and 5 mm from which the conical back plate is manufactured. This study will be useful to to the tool designer and the manufacturers doing work in this field*

Key Words: *Deep drawing, Digital environment, Finite element simulation, Forming, Manufacturing, Sheet metal, Tool designer.*

I. Introduction

Sheet metal forming is one of the extensively used manufacturing processes for the fabrication of a wide range of products in industries. Deep drawing is one of the most popular metal forming methods available to manufacturers. In deep drawing, a flat blank of sheet metal is shaped by the action of a punch forcing the metal into a die cavity. The use of numerical simulation could contribute towards the development and optimization of the process, leading to significant economic and technical gains. The application of the finite element method to the numerical simulation of the deep-drawing process has evolved in a significant way in the course of the last few years. Many of the problems associated with numerical simulation of this process have been solved or at least are better understood. Reviewing the various literature available on simulation of drawing, it is understood that most of the research work are focused on drawing of cylindrical product. Relatively a few research work has been done for finite element simulation conical product. The conical shaped product made on hydraulic power press were extensively used in the engineering and day today life e.g. in taper roller bearing, cookware, blowers of chemical, food and electrical industries etc..

According to Dr.Sc. Amra Tali ikmi et al,[1] deep drawing is a process for shaping flat sheets into cup-shaped articles without fracture or excessive localized thinning. Their paper describes the use of ABAQUS finite element code in a single stage sheet metal forming simulation on rectangle cup deep drawing. They suggested that the main objective of numerical simulation of the forming process is to reduce the development time of a new product. M. Afteni et al., [2] put forward that the increasing demands for small devices which have multiple applications in automotive industry, in chemical industry but also in medicine leads to new approaches concerning both the simulation and the experimental analysis of the material forming processes. They presented the experimental and numerical studies regarding the micro-deep drawing of Nickle sheets. In the study of Saad Theyyab Faris[3] , a numerical procedure was proposed for the design of deep drawing process using finite element method through program code ANSYS 5.4 simplified 2-D axisymmetric model of cylindrical cup are been developedEric T. Harpell et al., .[4] modelled various tooling geometries by using finite element analysis. Predicted strain distributions within drawn cups is assessed through comparison with code, LS-DYNA, for a cylindrical cup drawing proces. But in their study they worked on aluminium sheet.

L.F. Menezes et al. [5] presented a three-dimensional mechanical model for the numerical simulation of the deep-drawing process. The model takes into account the large elastoplastic strains and rotations that occur in the deep-drawing processAccording to Abdolhamid Gorji .[6] forming conical parts is one of the complex and difficult fields in sheet-metal forming processes. Simulation of elastic-plastic behavior of mild steel sheet is carried out with non-linear condition to gain accurate and critical understanding of sheet forming process by According to Laxmiputra M Nimbalkar et al. [7]

II. Process Description

During the process a piece of sheet metal is clamped between the die and the blank holder.A force is applied to the blank holder to prevent wrinkling of the sheet and to control the material flow during the deformation. When the punch is pushed into the die cavity the sheet deforms plastically and thereby it takes the specific shape of the tools.[8] An example of such a deep drawing part is given in figure 1.

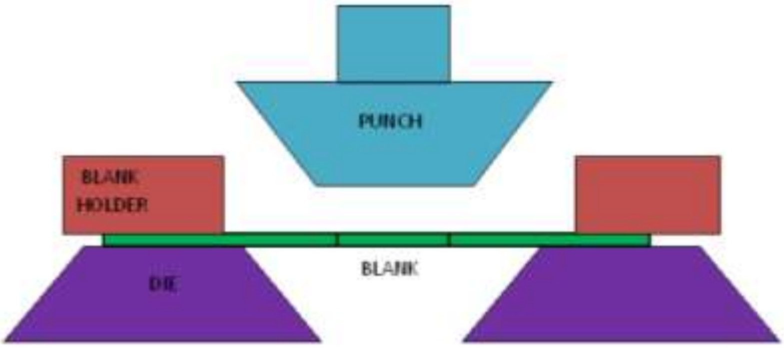

Fig. 1: Set up of a drawing tooling

III. Outline Of Finite Element Analysis

Finite element analysis is a simulation technique which evaluates the behavior of components, equipments and structures for various loading conditions. It is a computerized method for predicting how a real object will react to forces by mesh of simpler interlocking structures, the simpler structures or finite elements being agreeable to mathematical analysis. The finite element method is originally developed to study the stresses in complex aircraft structures. Then, it is applied to other fields of continuum mechanics, such as heat transfer, fluid mechanics, acoustics, electromagnetic, geomechanics, biomechanics. However, this paper is devoted solely to the topic of finite elements for the analysis of structures.

In the finite element analysis the structure is modeled by the assemblage of small pieces of structure, fig.2. These pieces with simple geometry are called finite elements. The word "finite" distinguishes these pieces from infinitesimal elements used in calculus. In the finite element analysis (FEA), the variation of the field variable on the element is approximated by the simple functions, such as polynomials. The actual variation on the element is almost certainly more complicated, so FEA provides an approximate solution. However, the solution can be improved by using more elements to represent the structure.

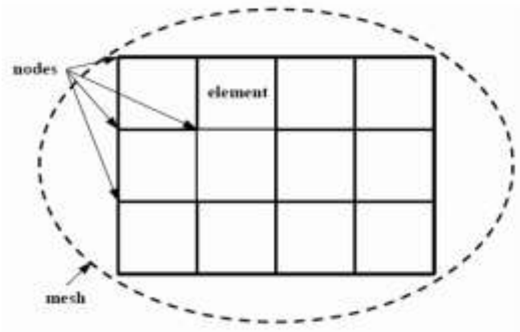

Fig. 2: Finite Element Mesh structure
(Source [9])

Elements are connected at points called nodes. The value of field variable and perhaps also its first derivatives are defined as unknowns at the nodes. The assemblage of elements is called a finite element structure, and the particular arrangement of elements is called a mesh. FEM changes the governing differential equations or integral expressions into a set of linear algebraic equations to solve the nodal unknowns.

Solving a structural problem by FEA involves following steps : Learning about the problem; Preparing mathematical models; Discretizing the model; Having the computer do calculations; Checking results (Generally an iteration is required over these steps.)

Finite element simulations are often required to reduce the experimental cost and time by reducing number of trials in the product development cycle. Metal forming is one of such area where a lot of trials are required to arrive at the die design to produce defect free parts. There is a body of literature that demonstrates finite element method can be used as a powerful analytical tool in simulation of deep drawing processes. If the finite element method is used properly in regards to reliable input data as well as correct modeling technique and appropriate analysis approach, then the results can be reliable enough to predict the outcome of drawing processes and thus direct the design of tools

IV. Methodology

A simple drawing situation is investigated both experimentally and by numerical simulation for four selected models of back plates. The blank from which these back plates are formed have the dimensions as as given in TABLE 1. The material of back plates is mild steel IS 2062 grade. A commercial FE code ANSYS 12.0 structural was used to simulate the deep drawing operation.

The punch is pushed into the die cavity, simultaneously transferring the specific shape of the punch and the die to the sheet metal blank, thus forming a conical cup. The basic shape of punch is frustum of a cone. The back plate is drawn from the blank in a press by the force of the punch. The experimental set up is shown in fig 8. Two different models of this experimental set up is analyzed. The geometrical detailing of these set ups are tabulated in table 1.The thickness of drawn portion is t_f.

Table 1 Parameters of experimental set up

Sr. No.	Description	Symbol	Model No.1	Model No.2
1	Top outer diameter of Punch	D_{PT}	450 mm	370 mm
2	Bottom outer diameter of Punch	D_{PB}	251 mm	206 mm
3	Top inner diameter of Die	d_{DT}	455.76	379.34 mm
4	Bottom inner diameter of Die	d_{DB}	256.76	215.34 mm
5	Blank Thickness before drawing	t_i	3.06 mm	5.05 mm
6	Depth of Drawing	h	67.25 mm	80.54 mm
7	Blank outer diameter	D_{BO}	930 mm	540 mm
8	Blank inner diameter	D_{BI}	214 mm	145 mm
9	Punch nose angle	α	30 degree	43 degree

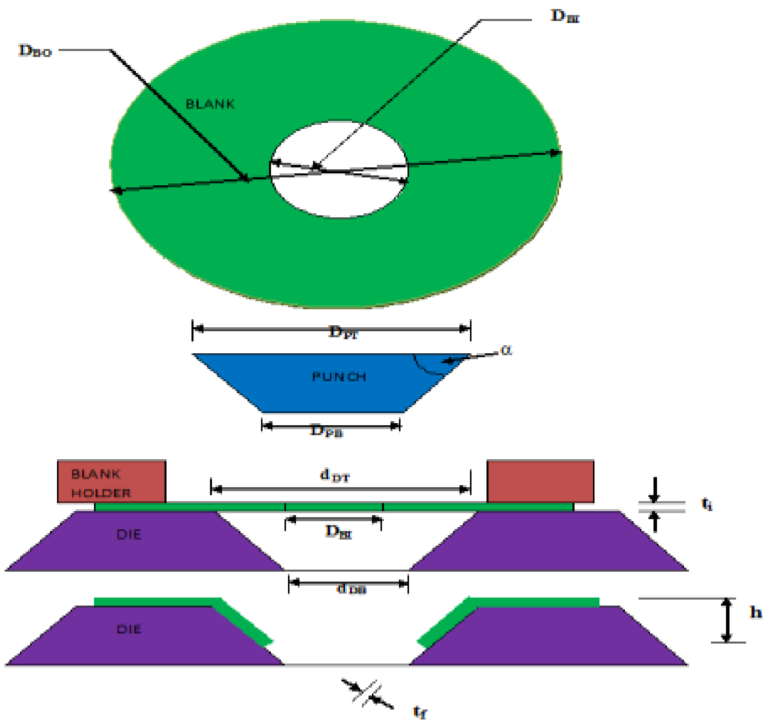

Fig. 3: Diagram of experimental set up

V. Simulation Procedure

5.1 Induction

A 2-D finite element model was created and suitable material properties were assigned to the model. Loads were applied as in the experimental tests to get numerical verification of the results. The various stages of this simulation work is summarized sequentially in the following steps :
Preprocessing: defining the problem; the major steps in preprocessing are given below:
Define key points/lines/areas/volumes (Solid Modeling)
Define element type and material/geometric properties
Mesh lines/areas/volumes as required
Solution: assigning loads, constraints and solving;
Apply the loads (point or pressure),
Specify constraints (translational and rotational)
Finally solve the problem.
Post processing: further processing and viewing of the results;
Lists of nodal displacements and show the deformation
Element forces and moments
Stress/strain contour diagrams

5.2 Solid Modeling

Solid Modeling can be defined as the process of creating solid models in CAD system. A solid model is defined by volumes, areas, lines, and key points.By using various geometrical data of presented setup a solid modeling of the same is created in ANSYS. Solid modeling of the experimental set up is done by using various dimension of the geometry as given in table 1.

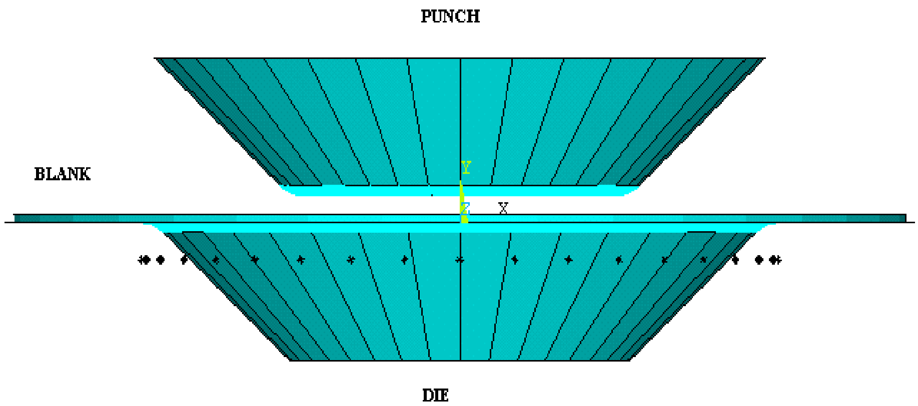

Fig. 4: ANSYS solid modeling of punch, blank and die

The geometry built in ANSYS is shown in fig-xxx. The structure is divided to ease map meshing of the problem. Work plane options are used to divide the structure. An axisymmetric approach is used to built the geometry. Axisymmetry is the best option to built and analyze deep drawing conical formations.

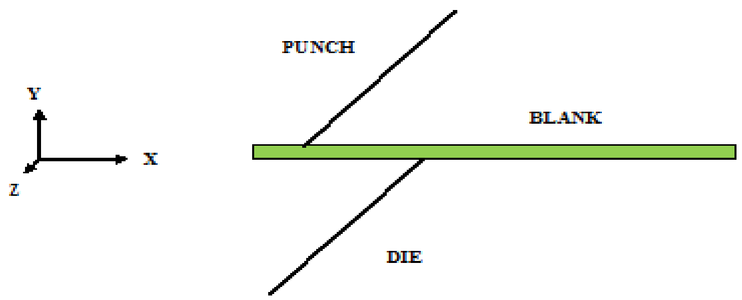

Fig. 5: ANSYS axisymmetric modeling of punch, blank and die

5.3 ELEMENTS

The type of element to be used in the analysis influences the exactness and accuracy of the results to a great extent. Literature review and examination of peer researchers' works show that PLANE42, 2-D elements with axisymmetric behavior have been conveniently used in the numerical analysis of axisymmetric forming process. This element is capable of representing the large deflection effect with plastic capabilities. This element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. The element is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. [10],[11],[12]

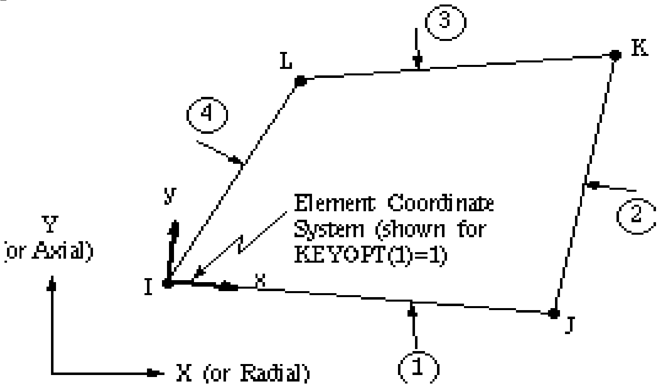

Fig. 6: PLANE 42, 2-D Structural Solid Element

Element size plays an important role throughout a simulation. Element size affect both the computation time and the accuracy of the results. The blank should be meshed finer enough in order to get acceptable results. However, the increase in number of elements results in a drastic increase in computational time. In the presented study in order to achieve good results, size of element is for model number 1 taken as 1.53 mm and for model number 2 it is taken as 2.525 mm.

5.4 Material Properties

The material of back plates is mild steel IS 2062 grade. It was selected as a structural , non-linear, isotropic hardening material model in the presented ANSYS simulation. Various material properties like yield stress, Modulus of elasticity, Poisson's ratio etc, which are required for fem simulation are obtained from various authentic literature. Tools are assumed as rigid, so there is no need to define material, however the material of punch and die is tool steel.

Table 2: Mechanical Properties of mild steel IS 2062 : source [13]

Sr. No.	Properties	Value
1	Tensile Strength	410 MPa
2	Yield Stress	250 MPa
3	Modulus of elasticity	200 GPa
4	Poisson's Ratio	0.3
5	Friction Coefficient	0.1

5.5 ANSYS MESHING

A quad mapped mesh was generated on all areas apart from the punch/die which is taken as rigid. This was done to achieve a higher number of elements along this line so a solution using contact conditions could be found easier. The figure 16 and 17 shows meshed model of the problem. 4 node PLANE42, 2-D elements is used to mesh the structure. The mapped mesh is good for accurate results as well as for graphical representation which is not proper with free mesh. An expansion option available with ANSYS is used to represent in the three dimensional space. Number of elements in the mesh for model number 1 and 2 are 468 and 158 respectively. Number of nodes for model number 1 and 2 are 1669 and 639 respectively.

Fig. 7: ANSYS Meshing For the Bank (Side View)

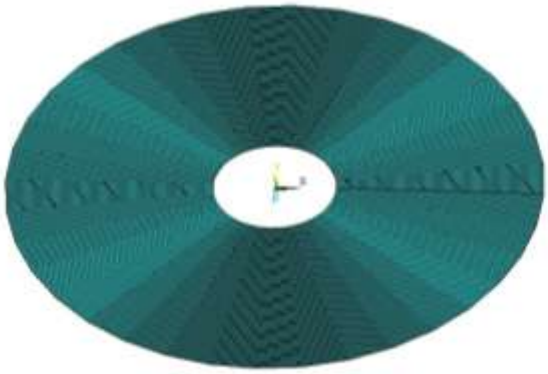

Fig. 8: ANSYS meshing for the bank (top view)

5.6 Loading & Boundary Conditions

As part of FE analysis, applying loads and constraints i.e. boundary conditions consists of defining which parts from geometrical model moves i.e. defining degree of freedom. Contact surfaces used in the presented work are top blank - bottom punch, bottom blank - top die. In current study movement of blank part is restricted in x- direction. Displacement load of the part of the blank which initially not in contact with die is given in y-direction. Movement of horizontal part of the blank which is on the die is restricted in x- direction as well as in y-direction both. The tools i. e. punch, die and blank holder, in finite element simulation are considered rigid because they are extreme stiff compared to the sheet. For this reason the tool can be presented as a surface only.

VI. Solution And Results

Following the successful modeling of geometry, then meshing and correctly applying boundary conditions and loads, a solution was run. The displacement load is applied and problem is executed in the nonlinear domain using material properties specified as in TABLE 2.

The solution was a large static displacement analysis. It was carried out with time increments, having the max number of sub steps set as 1000, minimum as 10, and a desired number of 100 specified. The strain and stress distribution was plotted on a contour plot to give a visual indication of strain through the now deformed blank.

The formations of sheet metal along with resulting strains are represented as shown in the following fig. 9 and fig. 10. From the color grid at the bottom of the fig. 9 and fig 10, the values of y-direction strains at the various points at the drawn component can be obtained. Fig. 9, contour plot for strain for model 1 and Fig. 10, contour plot for strain for model 2 shows the distribution of thickness strain in the drawn part of the blank. From the finite element simulation, the region of thinning can be identified. Higher strain regions as shown in fig 9 and 10.

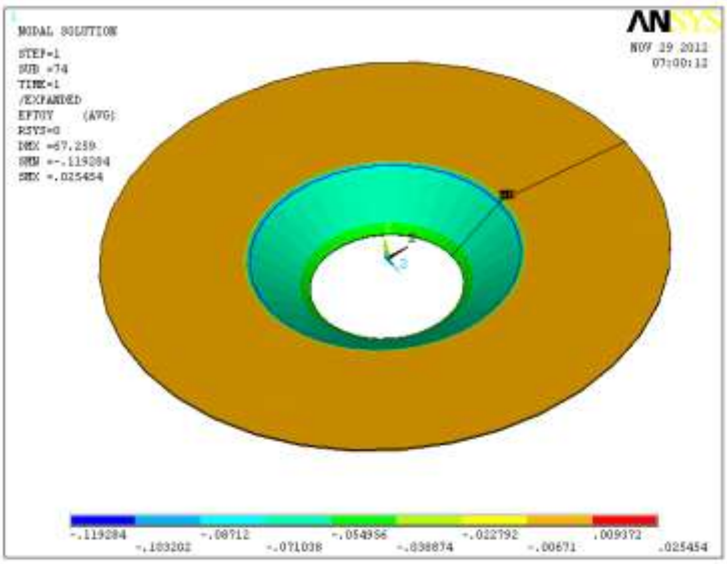

Fig. 9: Contour plot for strain for model 1

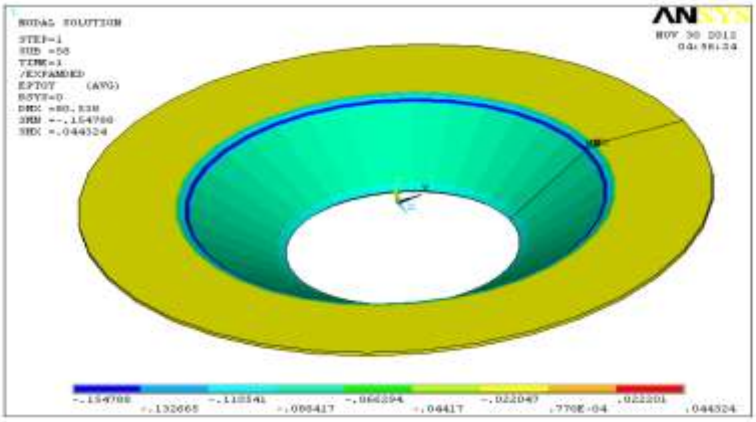

Fig. 10: Contour plot for strain for model 2

VII. Conclusions

In this paper, a method to simulate a drawing process in ANSYS was explained. In this simulation work the finite element analysis is done on the conical back plate of the impellers Over the course of this work 2 models were built and tested, these were:

Model No. 1: For M.S. Plate having thickness 3 mm, a 2-D Axisymmetric solid model built using all PLANE 42 elements, 30 degree punch nose angle used.

Model No. 1: For M.S. Plate having thickness 5 mm, a 2-D Axisymmetric solid model built using all PLANE 42 elements, 43 degree punch nose angle used.

Following results can be deduced from this effort:

(1.) With simulation via FEM, designers can estimate field variables such as strain distribution. This information enhances the design capability and knowhow of an experienced process designer and leads to a reduced number of die-tryout tests. We conclude that referring on space state of strain, the location of affected zones were established, which be helpful in punch/ die design and design of other parameters such as punch force, blank holder force etc

(2) The distribution of starin of the drawn component presented. The finite element model have proven to be in qualitative agreement with those of experiment.

(3) It is also concluded that ANSYS 12.0 is a very capable finite element software package, which can handle contact, plasticity and large deflection nonlinearities very accurately. Thus, forming processes can easily and accurately be modeled in ANSYS.

(4) It was concluded that, by using FEM, it is possible to produce successful conical shape products.

(5) By using a specialized software, one can be save time and other costs on research work. These simulation and analyses, presented here, suggests that, expensive way to find materials behavior by punch, die and experimental set up can be avoided by using specialized software.

However errors due to material properties may vary from coil to coil and affect the result. Finite element analysis determines an approximate solution of the problem modeled which have errors, these errors generally result from simplifications to the geometry, boundary conditions, material properties, loading and friction and as well as the discretization errors are a consequence of representing a continuum by a finite number of elements.

VIII. Suggestions For Future Work

Throughout the study it is observed that some of the areas can further be investigated and developed. In the presented study, the experiments were performed on the conical shape only. But the behavior of semi-circular, rectangular and triangular shape can be analyzed. In the current study the material is considered isotropic, a study can be developed by considering the anisotropy of the material. Material can be taken other than mild steel. A majority of products of automobile and aircraft industry are manufactured by drawing process. Any product can be taken for analysis work. Investigations on nonsymmetric workpieces can be conducted. Different element models can developed. Remeshing can be applied.

Reference

[1] Dr.Sc. Amra Tali ikmi, Muamer Trako, Mladen Karivan, Finite element analysis of deep drawing, 14th International Research/Expert Conference, Trends in the Development of Machinery and Associated Technology, TMT 2010A, September 2010, pp-11-18

[2] M. Afteni, M. Banu, V. Paunoiu, I. Constantin, Numerical and experimental investigations of the nickel thin sheets micro-deep drawing process, The annals of "dunărea de jos" university of galați fascicle v, technologies in machine building,ISSN 1221- 4566, 2011, pp 149

[3] Saad Theyyab Faris, Study of the stress and strain distribution during deep drawing and ironing process of metals with a circular profiled die, Diyala Journal of Engineering Sciences, ISSN 1999-8716, Vol. 02, No. 01, June 2009, pp. 80-95

[4] Eric T. Harpella,1, Michael J. Worswickb, Mark Finnc, Mukesh Jainc, Pierre Martind,Numerical prediction of the limiting draw ratio for aluminum alloy sheet, ELSEVIER, Journal of Materials Processing Technology 100 (2000), 2000,pp 131-141

[5] L.F. Menezes and C. Teodosiu, Three-dimensional numerical simulation of the deep-drawing process using solid fnite elements, ELSEVIER, Journal of material processing technology 97(2000), 2000,pp 100-106

[6] Abdolhamid Gorji & Hasan Alavi-Hashemi & Mohammad Bakhshi-jooybari & Salman Nourouzi & Seyed Jamal Hosseinipour, Investigation of hydrodynamic deep drawing for conical–cylindrical cups, Int J Adv Manuf Technol (2011) 56:915–927,DOI 10.1007/s00170-011-3263-0, Springer-Verlag London Limited ,2011,pp 915–927

[7] Laxmiputra M Nimbalkar, Sunil Mangshetty, Analyzing the Effect of Blank Holder Shape in Deep Drawing Process Using Fem, International Journal of Engineering Research and Development e-ISSN: 2278-067X, p-ISSN: 2278-800X,Volume 4, Issue 1, October 2012, pp. 23-28.

[8] I.Burchitz, Springback: improvement of its predictability, Project, NIMR project number MC1.02121, Strategic Research programme of the Netherlands Institute for Metals Research, March 2005, pp 1-13

[9] Serhat Yalçin, Analysis and modeling of plastic wrinkling in deep drawing, Thesis, Middle east technical university, September 2010,pp 31-34

[10] Hakim S. Sultan Aljibori, 2009, Finite Element Analysis of Sheet Metal Forming Process, European Journal of Scientific Research, Euro Journals Publishing, Inc. 2009,ISSN 1450-216X Vol.33 No.1, (2009), pp.57-69

[11] Dr. R. Uday Kumar, Finite element anlysis of evaluation of radial stresses in hydro -assisted deep drawing process, International journal of pure and applied research in engineering and technology, IJPRET, Volume 1(2):, ISSN: 2319-507X, 2012, pp 1-10

[12] GAO En-zhi, LI Hong-wei, KOU Hong-chao,CHANG Hui, LI Jin-shan, ZHOU Lian, 2009, Science direct, Transaction of non ferrous metals society of china, 19 (2009),pp 433-437

[13] IS 2062:1999, Indian standard specification code for Mild Steel, Indian standard,Steel for general structural purposes – specification (fifth revision),second reprint January 2002,pp 2-5