ISSN 2395-1621

Modeling, Analysis and Simulation of Crack and Thinning in the Sheet Metal Forming Process

[#1]Mr.Sharad Haridas Dethe [#2]Dr. Mangesh Phate

[1]dethesharad9@gmail.com
[2]mangesh_phate@rediffmail.com

[#1]Student of M.E (Design Engineering), PVPIT Bavdhan, Savitribai Phule Pune University, India
[#2]Department of Mechanical Engineering, PVPIT Bavdhan, Savitribai Phule Pune University, India

ABSTRACT

In the 21st century sheet metal simulation technology is omnipresent in day to day life of a booming industrial sector. In the development of sheet metal forming, momentary trial and error technique was employed to determine permanent plastic deformation resulting in the material properties dissimilarity. Therefore it is necessary to determine the extent to which a material can further be deformed for subsequent forming. This approach is time consuming and depends heavily on the experimentation and trial. To concentrate on these limitations, sheet metal forming simulations has been applied to detect crack and thinning in sheet metal forming process. In this paper simulation is carried out on Ansys and Hyper form software's. Simulation result validation is carried out by experimentation process. By using sheet metal simulation we can easily analyse the meshing conditions, stress analysis and failure point in sheet metal. After analysis and simulation of CAD model we can easily finalize the forming die design, forming process parameter and experimentation which is the motive of this paper.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

***Keywords*—** **Crack and thinning, Sheet metal forming, Simulation and Validation.**

I. Introduction

Sheet metal forming is the process in which force is applied to a piece of sheet metal to change its geometry without removing any material. Metal forming typically faces difficulties like wrinkling, thinning, crack and optimization of shapes. All of which are expensive as they lead to wastage of material and loss of production time. Simulation can predict such defects during product development and often avoid their happening during production with the assistant saving of time and material, by identifying necessary and often times simple changes in design. It is of importance for sheet metal fabricating companies to adopt product development simulation techniques in order to reduce or eliminate the aforementioned problems and thus remain competitive.

Sheet metal forming is one of the most widely used manufacturing processes in industry that is used to change the geometry of sheet metal. Sheet metal forming is used to produce various products from stainless steel, copper, aluminium, nickel, brass and titanium. To reduce costs and increase the performance of designed products, more and more lightweight and high strength materials have been used. These materials usually have limited formability. Thus a thorough understanding of deformation processes and the factors limiting the forming of sound parts is important from both engineering and economic viewpoints.

In sheet metal forming operations, the entire of useful deformation is restricted by the occurrence of uneven deformation which mainly takes the form of necking. Wrinkling occurs when the dominant stresses are compressive, tending to cause thinning of the material. Localized necking occurs when the stress state leads to an increase in the surface area of the sheet at the outflow of a reduction in the thickness. The two kinds of neck that occur are diffuse necking and the localized necking (through thickness), which is concluded by final separation or crack. After the localized neck initiates, auxiliary deformation of the material concentrate in this localized region, and homogeneous deformation away from neck region vanishes completely. The localized neck is therefore a very important phenomenon in determining the amount of useful deformation that can be imposed on a work piece. The mechanism for initiation of localized band involves a number of factors including material properties and punch profile.

In this paper three different blanks are chosen with initial thickness of 1.00mm, 1.20mm & 1.50mm with blank outer diameter of 65mm. The material thickness is optimised by using stress concentration intensity in simulation [8].

In this modeling is carried out on Ansys, Solid Edge ST (104.00.00.082) Femap (10.2.1), Solver used NX Nastran (7.1) and Hyper form. Simulation result validation is carried out for thinning and crack in sheet metal forming by experimentation process. By using sheet metal simulation we can easily analyse the meshing conditions, stress analysis & failure point in sheet metal which is the motive of this dissertation [5].

II. LITERATURE REVIEW

Now a day's simulation in the automobile industry is an imperative to the sheet metal forming process. Simulation used today is the result of a development process through the last two decades. A main focus of this process was the continuous optimization to characterize the material properties [1], [5].

Single point incremental forming (SPIF) is a new innovative and feasible solution for the prototyping and the manufacturing of sheet metal. In incremental Sheet Metal Forming the blank is step wise curved in to a desirable shape by tool punch passing through die block [7].

This paper explore about simulation method in Ansys and Hyper form software. Simulation result validation is carried out for crack in sheet metal forming by experimentation process. With the help of simulation the meshing, stress analysis & failure modes in sheet metal forming are without difficulty analysed.

III. METHODOLOGY

The importance of sheet metal working process in present technology is due to the ease with which metal may be formed into useful shapes. Deep drawing can be done in single draw or double draw. When it is completed in double draw then, it is called re-drawing or restrike operation. Number of draws can be calculated by taking ratio of blank diameter to the required diameter of the cup which is known as Draw Ratio (DR). In this paper three different blanks are chosen with initial thickness of 1.00mm, 1.20mm & 1.50mm with blank outer diameter of 65mm [8].

A. *Flow Chart and Methodology:*

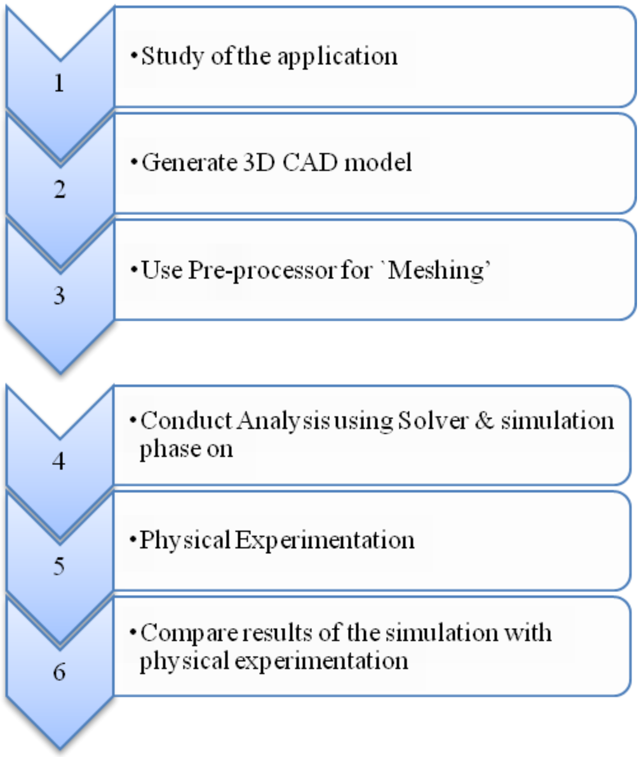
Methodology of this simulation work starts from the 3D CAD model generation in ANSYS. This CAD model is used further for meshing, analysis and for simulation purpose in HYPER FORM. 3D Cad model can be generated by the majority of approaches used to date still follow the traditional CAD route by using an intermediary step of surface reconstruction which is then followed by a traditional CAD-based meshing algorithm. CAD-based approaches use the scan data to define the surface of the domain and then create elements within this defined boundary [10].

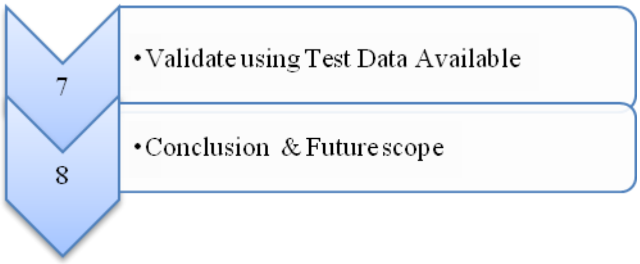
In this paper we have circular CAD model (called as Retainer or holder in industrial definition). The FEA consists of three steps: Pre-processing, Processing (or Solution) and Post-processing. A complete finite element analysis is a logical interaction of these three steps [12].

B. *Pre-processor in Meshing*

Pre-processing is the primary step in FEM. It includes geometry creation, meshing and applying boundary conditions. The software ANSYS or HYPERMESH includes tools to create geometry in itself, but it is preferred to create 3D model in CAD software. Then model is to be imported in the analysis software to create meshing. Meshing is the process of discretization of the geometry into elements and

After meshing the suitable boundary conditions are to be applied like Pressure, force, displacement, fixed support etc. In general we can say that pre-processing is the step of creating a deck for the analysis [15].





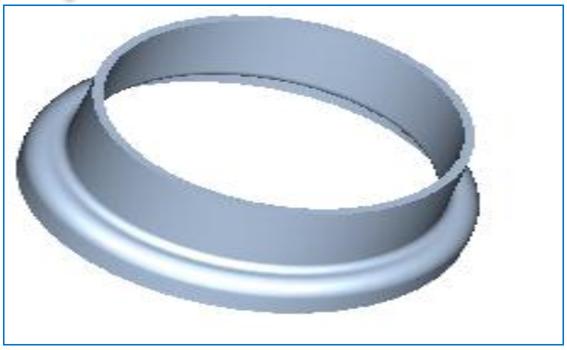


Fig. 2: 3D CAD Model of Retainer

C. Draw operation calculations by Conducting Dynamic Analysis using Solver:

1) Type of operation: [9]

In sheet metal forming operations, the amount of useful deformation is limited by the occurrence of unstable deformation which mainly takes the form of localized necking or wrinkling. Failure by wrinkling occurs when the dominant stresses are compressive, tending to cause thickening of the material. Localized necking occurs when the stress state leads to an increase in the surface area of the sheet at the expense of a reduction in the thickness. The two kinds of neck that occur are diffuse necking (so called because its extension is much greater than the sheet thickness), and the localized necking (through thickness thinning), which is terminated by final separation or fracture. After the localized neck initiates, further deformation of the material concentrates in this localized region, and homogeneous deformation away from neck region vanishes completely [5]. The localized neck is therefore a very important phenomenon in determining the amount of useful deformation that can be imposed on a work piece. The mechanism for initiation of localized band involves a number of factors including material properties and punch profile. The phenomenon is attributed to the softening effect, including geometric softening (the decrease with strain of the cross-section area which bears the forming load, the generation of voids), and material softening (flow stress decreases with the increase of the effective strain).

$h / d \leq 0.5$ – shallow drawing ... (1.1)
$h / d > 0.5$ – deep drawing
Where, $h/d = 23.50/45.4=0.6$ ... (Deep drawing operation)
h = shell height
d = shell diameter

2) Estimation of blank Diameter (Theoretical): [9]

$D = \sqrt{d^2+4dh - 0.5r}$ ... (1.2)
$= \sqrt{45.4*45.4+445.4*17.00 - 0.5*3.00}$
$= \sqrt{45.4*45.4+445.4*17.00 - 0.5*3.00}$
= 65.00mm ... (Blank diameter)

Where,
D – Blank diameter in mm
d – Shell outer diameter in mm
h – Shell Height in mm
r – Corner radius of punch
When d / r is between15&20

3) Considering Trim allowance: [9]

Trim allowance = 0.005mm for every 10 mm diameter of drawn cup … (1.3)

Where,
D1= Initial diameter of blank (D1)
= D (Theoretical diameter.) + Trim allowance
= 65.00+0.005*45.40 = 65.224mm

4) T / D Consideration: [9] Following results of wrinkling are obtained from sheet thickness to blank diameter ratio on the basis of simulation in hyper mesh.

Table I: T / D decides the severity of wrinkling

T / D (%)	Severity of wrinkling
Up to 0.5	Wrinkling is a rigorous and compressive load must be reduced. Blank holder must be used, so a double action press is preferable
0.5 to 1.5	Wrinkling is moderate and low blank holding forces are permitted
1.5 to 2.5	Wrinkling is very light so, single action press is enough

5) Selection of percentage reduction for 1st Draw using t / D ratio: [9]

Table II: (t / D) x 100 % reduction for 1st Draw

(t / D)x100		% reduction for 1st Draw [(D1 – D2) / D1] x 100
Single action	**Double action**	
1.5	0.15, 0.2, 0.30	30, 35, 40

Where,
D1 = Blank diameter. After adding trim allowance
D2 = Diameter. Of 1st Draw
And Allowable percentage reduction for successive draws
Case a) t=1.00mm, *Case b)* t=1.20mm
First draw (t / D) % = 1.00/65.22= 1.53
First draw (t / D) % = 1.20/65.22= 1.84
Second draw 25% of reduction required for this thickness
Second draw 25% of reduction required
Third draw 15%, not required
Case c) t=1.50mm
First draw (t / D) % = 1.50/65.22= 2.30
Second draw 45% of reduction required with double action for this thickness
Third draw 15%, not required

6) Estimation of drawing pressure: [9]

$P = \pi \times d \times t \times S \times ((D / d) - C)$ … (1.4)

$= 3.14*48.40*1.20*505*(65.22-48.40) - 0.62$

$= 3.10$ Kg-f

Where, P = Drawing force in 'kg-f'

d = Shell outer diameter

D = Blank diameter

t = thickness of sheet in 'mm'

S = Ultimate tensile strength in N/mm2 (505Mpa)

C = constant to cover friction and bending (0.6 to 0.7 for ductile material)

7) Blank holding pressure: [9]

Blank holding pressure = 1/3rd of drawing pressure

= 3.10/3= 1.03Kg-f … (1.5)

8) Press capacity: [9]

Press capacity = (Drawing pressure + Blank holding pressure) x 1.3 = (3.1+1.03) x1.30 … (1.6)

= 5.369 Kg-f

9) Forming speed: [9]

Table III: Forming speed

Material	**Single action drawing**		**Double action drawing**	
	Ft/min	**m/sec**	**Ft/min**	**m/sec**
Steel	60	0.3048	30-55	0.1778-0.27
Stainless Steel	-	-	20-30	0.1016-0.27

10) Material properties of 304SS: [9]

IV. Simulation, Experimentation & Validation

A) Simulation:

In metal forming simulation, the forming of sheet metal is simulated on the computer with the help of Ansys, Solid Edge ST (104.00.00.082) Femap (10.2.1), Solver used NX Nastran (7.1) and Hyper form. Simulation result validation is carried out for thinning and crack in sheet metal forming. Simulation makes it possible to detect errors and problems, such as wrinkles or splits in parts, thinning or crack on the computer at an early stage in forming. In this way, it is not necessary to produce real tools to run practical tests. Forming simulation has become established in the automotive industry since it is used to develop and optimize every sheet metal part.

To illustrate the metal forming process, there must be a model of the real process. This is calculated in the software using the finite element method based on implicit or explicit incremental techniques. The parameters of the model must describe the real process as accurately as possible so that the results of the simulation are realistic.

Study Property	Value
Study name	Dynamic & Static Study 1
Study Type	Linear
Mesh Type	Tetrahedral
Iterative Solver	On
NX Nastran Geometry Check	On
Surface results only option	On

Material Property 304ss	Value
Density	8027.000 kg/m^3
Coef. of Thermal Exp.	0.0000 /C
Thermal Conductivity	0.017 kW/m-C
Specific Heat	502.000 J/kg-C
Modulus of Elasticity	193053.196 MegaPa
Poisson's Ratio	0.290
Yield Stress	255.106 MegaPa
Ultimate Stress	579.160 MegaPa
Elongation %	0.000
Mesh type	Tetrahedral
Total no. of bodies meshed	1
Total number of elements	10,927
Total number of nodes	19,686
Subjective mesh size (1-10)	10

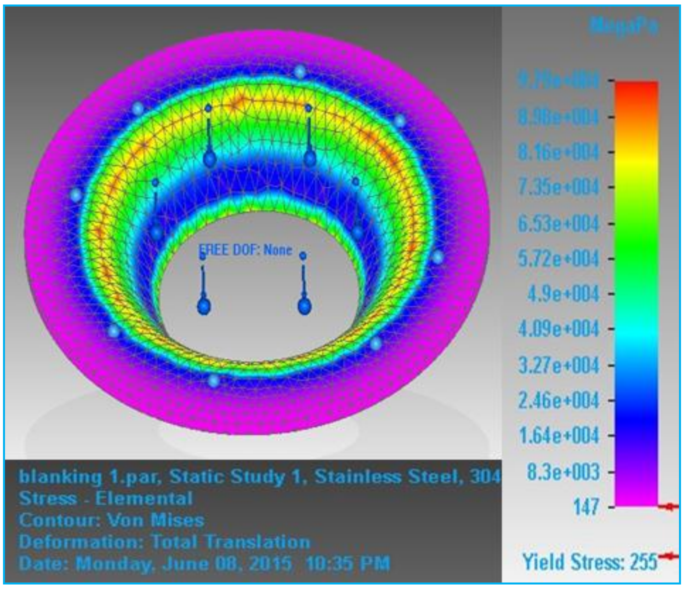


Fig. 3: Stress Result on component: Von Mises

Stress Result on component: Von Mises				
Extent	**Value**	**X**	**Y**	**Z**
Min	0.0096 MegaPa	-3.85 mm	-30.75 mm	-1.50 mm
Max	7.43 MegaPa	-18.57 mm	-10.47 mm	-0.10 mm

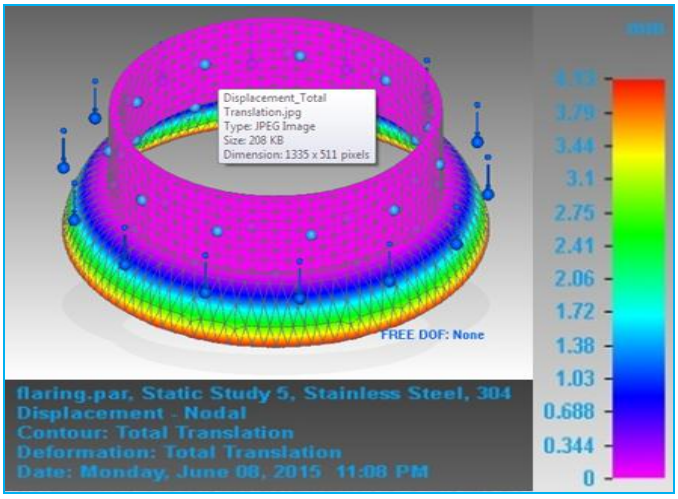


Fig. 4: Stress Result on component: Von Mises

Stress Result on component: Von Mises				
Extent	**Value**	**X**	**Y**	**Z**
Minimum	0.0007 MegaPa	24.20 mm	0.00 mm	14.80 mm
Maximum	8.49e+004 MegaPa	-24.23 mm	19.32 mm	1.50 mm

B) Experimentation: Experimentation will be conducted on a hydraulic press of a suitable capacity. Hydraulic press of 45T capacity with cushion pins selected for this forming operation [9]. 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 & simulation data. 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 draw ability of steel. Drawing has become a highly technical process, and the development of a steel forming route no longer involves simple trial and error methods.

The parameters influencing the draw operation are:

a) Type of material and Thickness of the material
b) Mechanical properties (Limiting Draw Ratio)
c) Blank holding pressure
d) Speed of the operation


Fig. 5: Hydraulic press (Capacity 45T)

Fig. 6: Experimentation die of forming or draw or flaring process

Fig. 7: Forming or draw die open view from top side

C) Validation: Validation of this simulation data carried out as, the appropriate capacity press selected by knowing the drawing load calculated by equation (1.4) and (1.5). Working with the presses of higher capacities may lead to many types of defects such as cracks and tearing. Blank holder pressure needs to optimize over a given range for optimized geometry.

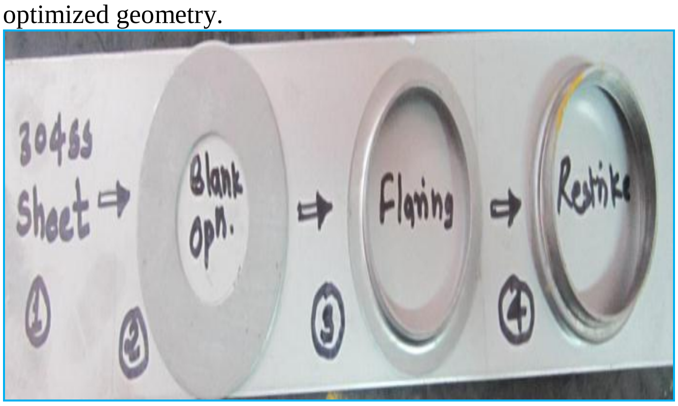


Fig. 8: Validation of methodology or forming process

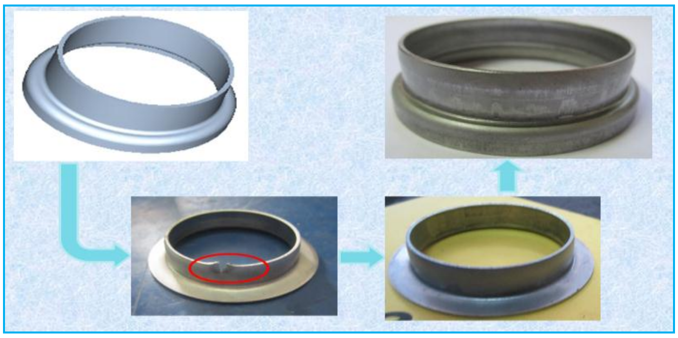


Fig. 9: Experimental Validation of crack

Fig. 9: Experimental Validation of thinning

The coefficient of friction needs to be optimized for the new geometry. Generally the deep drawing objects are analysed for their strength and failures with circle grid analysis, which is practically carried out on a sample piece, which is known as formability analysis. Alternatively, the actual trials performed over the component would directly reflect over the ease of drawing operation offered for the said simulation data in hyper form.

Validation result can directly measured practically by two methods, first is by observing visually for crack and secondly by measuring thinning of material by point vernier scale.

V.CONCLUSION

To overcome the entire problem related to thinning & crack in sheet metal forming simulation realized follows;

a) Simulation & Finite Element Method is easy to use and user friendly methods in sheet metal forming crack analysis, simulation & optimization.

b) Use simulation reduces time as well as production & development (trial & error) cost in sheet metal.

c) Through all above references available, simulation appears to be an efficient choice for optimization of the forming.

VI.PROPOSED FUTURE SCOPE

1) To do simulation of the actual sheet metal component over cad model with crack detection in simulation phase for material thickness 0.80mm to 1.50mm.

2) To do the physical experimentation & validation of the simulation & analysis results.

3) To do analysis of fracture & wrinkle defect in forming.

REFERENCES

[1] M. Firat. Computer aided analysis and design of sheet metal forming processes: Part III: Stamping die-face design, Materials and Design 28 (2007) 1311–1320. 23 March 2006.

[2] M. Firat. Computer aided analysis and design of sheet metal forming processes: Part II – Deformation response modeling, Materials and Design 28 (2007) 1304–1310.

[3] Rajiv Shivpuri, Wenfeng Zhang. Robust design of spatially distributed friction for reduced wrinkling and thinning failure in sheet drawing. journal homepage: www.elsevier.com/locate/matdes Received 30 July 2008 Accepted 30 August 2008 Available online 12 September 2008

[4] Z.Q. Sheng, S. Jirathearanat, T. Altan. Adaptive FEM simulation for prediction of variable blank holder force in conical cup drawing. International Journal of Machine Tools & Manufacture 44 (2004) 487–494. Received 20 April 2003; received in revised form 25 September 2003; accepted 5 November 2003. www.elsevier.com/locate/ijmatool

[5] Karl Roll (2008) “Simulation of Sheet Metal Forming – necessary developments in the future”, LS-DYNA Anwenderforum, Bamberg.

[6] Malwad, D. S. and Nandedkar, V. M. (2014) “Deformation Mechanism Analysis of Single Point Incremental Sheet Metal Forming”, Procedia Material Science Vol. 6, pp. 1505 – 1510.

[7] Suresh, K., Regalla, S. P. (2014) “Effect of mesh parameters in finite element simulation of single point incremental sheet forming process”, Procedia Material Science Vol. 6, pp. 376 – 382

[8] Sponsorship of this project is given by AESSEAL INDIA PVT LTD, Pune. Problem definition is defined by this sponsored company to resolve the sheet metal thinning & crack issue with the help of modelling, simulation, new die design & development with reference to the simulation in forming operation.

[9] 3331 ch7-Design of press working tools book http://facweb.spsu.edu/met/dhorton/3331/Fundamentals%20of%20Tool%20Design,3rd%20Edition/3331%20ch7-Design%20of%20press%20working%20tools.pdf

[10] Anup K. Sharma, Dinesh K. Rout. Finite element analysis of sheet Hydro mechanical forming of circular cup. journal homepage: www.elsevier.com/locate/jmatprotec

[11] Thipprakmasa, S., Rojananan b, S., Paramaputi, P. (2008) “An investigation of step taper-shaped punch in piercing process using finite element method”, Journal of materials processing technology Vol. 197, pp. 132 – 139.

[12] Chou, C. H. and Pan, J. (1994) “Analysis of sheet metal forming operations by a stress resultant constitutive law”, International journal for numerical methods in engineering Vol. 37, pp. 717 – 735.

[13] Kim, H. S., Koc, M., Ni, J. (2008) “Development of an analytical model for warm deep drawing of aluminum alloys”, Journal of materials processing technology Vol. 197, pp. 393 - 407.

[14] Jouna, M., Kima, M., Kimb, J., Chung, W. (2014) “Finite element analysis of deep piercing process”, Procedia Engineering Vol. 81, pp. 2494 – 2498.

[15] Vorkov, V., Aerens, R., Vandepitte, D., Duflou, J. R. (2014) “Springback prediction of high-strength steels in large radius air bending using finite element modelling approach”, Procedia Engineering Vol. 81, pp. 1005 – 1010

[16] Taylor, L., Cao, J., Karafillis, A. P., Boyce, M. C. (1995) “Numerical simulations of sheet metal forming”, Journal of materials processing technology Vol. 50, pp. 168 – 179.

[17] Hagenah, H., Geiger, M., Merklein, M. (2013) “Numerical simulation to improve robustness in sheet

metal forming process chains", Proceedings of AIP conference.

[18] Hartel, S., Awiszus, B. (2014) "New processing technologies of incremental sheet metal forming", Procedia Engineering Vol. 81, pp. 2311 – 2317.

[19] Balasubramanian, M., Ramanathan, K., Senthil kumar, V. S. (2013) "Mathematical modelling and finite element analysis of superplastic forming of Ti-6Al-4V alloy in a stepped rectangular die", Procedia Engineering Vol. 64, pp. 1209 – 1218.

