Available online at www.ijapie.org

INTERNATIONAL JOURNAL OF ADVANCED PRODUCTION AND INDUSTRIAL ENGINEERING

IJAPIE-2017-02-215, Vol 2 (2), 65-70

IJAPIE
Connecting Science & Technology with Management.

A Journal for all Products & Processes.

Design and Analysis of Progressive Die Using Finite Element Method

Dinesh Kumar Kushwaha[1], S.S Banwait[2], Husain Mehdi[1]
[1]*Department of Mechanical Engineering, Meerut Institute of Technology, Meerut, U.P, India*
[2]*Department of Mechanical Engineering, NITTTR, Chandigarh, Panjab, India*
**Email: husainmehdi4u@gmail.com*

Abstarct : *Historically, Early man had made use of wooden die to carve bricks. Since then, dies had been used in industries, especially for sheet métal and other forming operations. Progressive dies are forming tool comprises of two or more stations performing different operations. The raw material travels from first station to last to get transformed into the finished product. In this work an industrial problem at Anand Tracks and Field (P) Limited, regarding the shell manufacturing of discus throw was identified. The problem in shell manufacturing was regarding the concentricity of shell. The shell after being manufactured was not found exactly concentric; it means that the shell show some degree of straightness at the ends. Welding operation was then used to join the two ends so as to ensure the concentricity of the shell which, consequently increased the manufacturing lead time as well as labor cost. The problems encountered in the company was solved by designing a progressive die and it was suggested to change the raw material, instead of rod, plate was proposed to produce the shell. In this work, three stages of die design had been done. In the first phase, conventional designs had been done and the induced stresses were checked out. In second phase, the die and punch assembly was developed by using CATIA V5 R 20 software. Eventually, the drawing had been imported in ANSYS 14.0 software; a finite element method analysis software to check the stresses induced*

Keywords: *Stress, Progressive Die, Finite Element Method*

1. INTRODUCTION

The use of an automated progressive design system with multiple operations such as piercing, bending and deep drawing for manufacturing products is developed. Especially a approach to make a progressive and flexible working system which was based on knowledge base rule [1]. In order to reduce the stress on the tool during punching/blanking operation the use of finite element technique is made. Analysis of punching/ blanking force is done by varying the models of punching and blanking tools by using finite element models. This model enables the analysis of effects of variations of tool geometry on punching/blanking force and on the deformation of punch [2]. Industrial application of dies involves structural design tool for punches and dies of motor core based on the functional component. The descriptive model of punch and dies was established which contains three aspects of information, geometry information, assembly related constraint attribute and hole related information on plate [3].The optimization of the punch–die clearance values for a given sheet material and thickness by using the finite element technique and Cockroft and Latham fracture criterion. In the study, the shearing mechanism has been studied by simulating the blanking operation of an aluminum alloy 2024 [4]. A model to predict the shape of cut size in the sheet metal components has been established. The model investigates the effect of potential parameters influencing the blanking process and their interaction. This helped in choosing the process leading parameters for the two identical products manufactured with a two different materials blanked with a reasonable quality on the same mould. Finite Element Method and Design of Experiment (DOE) is used in order to achieve the intended model objectives. The combination of both the technique is proposed to result in a reduction of necessary experimental cost and effort in addition to getting a higher level of verification [5]. The blanking of thin sheet (0.1–1 mm) of mild steel using an elastic–plastic finite element method based on the incremental theory of plasticity. The blank diameters are considered in the range 1–4 mm. The study has helped to evaluate the influence of tool clearance, friction, sheet thickness, punch/die size and blanking layout on the sheet deformation. The punch load variation with tool travel and stress distribution in the sheet have been obtained [6]. The behavior of the blank material during the process through finite element simulations, analytical modeling and experimental work. The objective is to contribute towards the development of a system for the on-line characterization of the blank material properties during the punching/blanking process [7].The simulation of fine blanking processes, and a finite element model valid for numerically describing of such operations has been developed. The numerical simulations of the damage evolution and crack initiation and propagation have been described by means of continuum damage approach [8]. The finite element (elastic-plastic) analysis of sheet metal forming process using the finite element software. LUSAS simulation is carried out to gain accurate and critical understanding of sheet forming process. Axi-symmetry element mesh and plain strain element mesh were use incorporated with slide-line features to model and study the sheet metal forming process. Simulation of elastic-plastic behavior of aluminum sheet was carried out under non-linear condition to investigate sheet metal forming process [9]. Further the application of dies in manufacturing concern incorporates the methodology for progressive die sequence design for forming round cups using finite element method (FEM) based simulations. The process sequence design developed has been applied to forming of an automotive part and is compared with the design obtained from past experience. The methodology proposed that the integration of design experience and FEM simulations could enhance the robustness of the procedure for die design sequence and reduces the die development cost considerably [10]. The shearing mechanism is studied by simulating the blanking operation of an AISI 304 sheet and Simulation used the FEM program ANSYS. Also a methodology to obtain optimum punch-die clearance values for a given sheet material and thickness to be blanked, using the finite-element technique is developed [11].Again Finite element analysis to investigate the effect of the die clearance on shear planes in the fine blanking of a part of an automobile safety belt. For the analysis, AISI1045 is selected as the material, which was used in manufacturing part of an automobile safety belt [12]. More over simulation of sheet-metal cutting processes by shearing mechanisms, such as blanking and punching have developed a finite element model (FEM) valid for the numerical description of such processes. To study the effects of varying the process parameters on the geometry of the sheared edges, and the evolution of the force-punch penetration, they implemented a calculation algorithm by means of the user routine (UMAT) of ABAQUS/standard finite element code [13].Another industrial problem involves cutting of sheet-metal cutting processes by shearing mechanisms, such as blanking and punching, and developed a finite element model (FEM) valid for the numerical description of such processes. To study the effects of varying the process parameters on the geometry of the sheared edges, and the evolution of the force-punch penetration, they implemented a calculation algorithm by means of the user routine (UMAT) of ABAQUS/standard finite element code [14].

2. PROBLEM FORMULATION

M/s Anand Track and Field Equipment Private Limited is a manufacturer of various types of sports goods and Athletics & Gymnastic Gym Equipments. In one of its vendor unit, M/s Kaushik Engineering Works which is manufacturing the shell of discus throw; a problem has been identified by the management of the company regarding the manufacturing of shell. The shell after being manufactured is not found exactly concentric; it means that the shell show some degree of straightness at the ends as shown in fig.1.

Fig: 1 Semi finished Shell

3. RESULTS AND DISCUSSION

The straightness of ends of shell was the major problem in manufacturing. In order to maintain the concentricity, a number of miscellaneous operations were carried out which increased the manufacturing lead time as well as the labor cost. If the operations are combined or eliminated it will surely reduce the manufacturing lead time as well as labor cost. A progressive die and punch is the proposed possible solution for the problem encountered by the workmanship.

3.1 DESIGN OF DIEASSEMBLY

Progressive dies are made with two or more stations arranged in sequence. Each station performs an operation on work piece or provides an idler station, so that the work is completed when the last operation has been accomplished. In this work, as a solution to non-concentricity of shell, the raw material as well as the entire manufacturing processes was changed. In this new solution, the raw material recommended for shell manufacturing is mild steel plate and shell was carved out from a two station progressive die. The plate was first punched under smaller die and is then transferred to larger one and finally to the cavity dies which are already available in the industry. After having passed from the dies, a single piece shell was carved out.
In conventional design, taking into account the above dimensions of the shell of progressive die, punch was designed and the force on the punch was calculated. The material recommended for making the dies for the industrial application is H13 steel. Die steel H 13 was selected for manufacturing of the die and punch assembly. The die assembly for shell manufacturing is shown in fig 2.

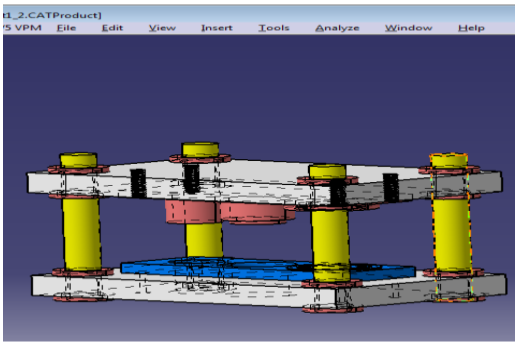

Fig: 2 Punch and Die Assembly.

The dimension of the punch holder which is comprises of three punches and dimensions are shown in fig 3. .

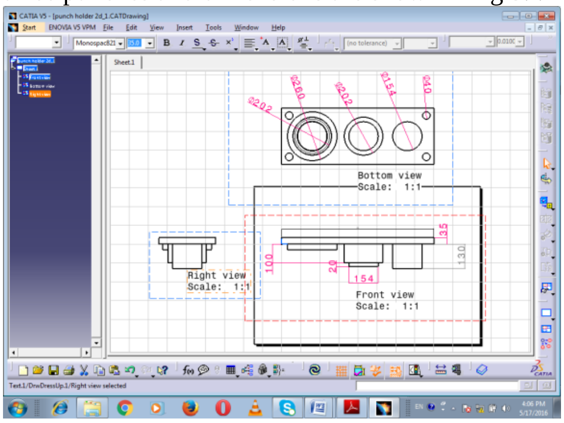

Fig : 3 2D View of Punch Holder

3.2 CALCULATION OF PUNCHING FORCE

For the calculation of force of punching, a standard formula is used and the maximum shearing force required to cut the sheet is

F_{max}= 0.7x UTS x T x L

Where

T= thickness of the sheet.

L= π x D= total length of sheared area.

{Taking UTS = 432-510 MPa.(C 20 carbon steel)}

F_{max}= 0.7x UTS x π x $D_{b\,x}$ T

F_{max}= 0.7x UTS x π x T x D_l

$F_{max\,=}$ 0.7x 510x π x 19 x 202.

$F_{max\,=}$ 4.5 MN.

3.2.1 Design of Punch Holder

The punch holder holds all the punches and the required load for punching is applied on the punch holder only. The material used for punch holder is H 13 steel. The punch holder has three punches one is smaller, second one is larger and third one is cavity punch as shown in fig 4. The cavity punch is used to give desired shape to the shell. As low magnitude of force act on cavity punch so in design it was neglected.

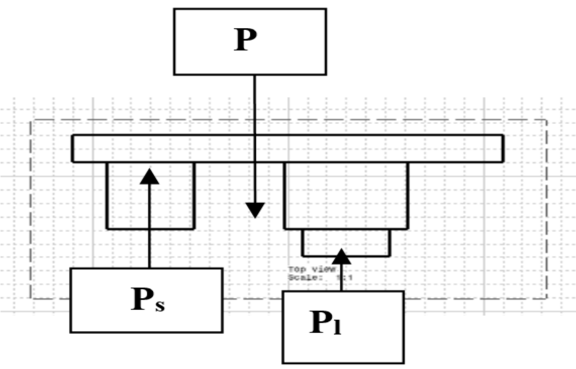

Fig: 4 Front View of Punch Holder

The magnitude of P is taken as 4.5 MN in one case and 12.5 MN in the another case to design the punch holder. Taking both the cases for calculating the force on punches and the induced corresponding stresses.

Case I: In this case, P = 4.5 MN and the sheet was first clamped on the smaller station and then is made to travel to larger station after the operation on first station was completed and then, it is finally clamped to the last station where only the final shape is given to the shell. In this case, the power requirement was less as compared to the second case. The sheet was first subjected to compressive strength and then shearing of sheet commences. Hence, the sheet was designed in compression. The dimensions of the punches are as follows:

D_s= 154 mm, D_l= 202 mm.

Checking for induced stress

The induced compressive stress in sheet in the smaller punch and in bigger punch is different as the area which is resisting the load is different and is calculated as under. The induced compressive stress is

$\sigma_{cs=}\ \frac{P}{\frac{\pi}{4} x\, D_s^{\,2}}$ = 4.5 x 10^6/ 0.785 x 154^2 (1)

Where,

$\sigma_{cs=}$ Stressesinduced in smaller punch.

$\sigma_{cs\,=}$ 241.59 MPa

$\sigma_{cl=}\ \frac{P}{\frac{\pi}{4}\,(D_l^{\,2}-D_s^{\,2})}$ = 4.5 x 10^6/ 0.785 x (202^2 – 154^2)

σ_{cl} = 4.5 x 10^6/13.41 x10^3 = 335.57 MPa

Where

$\sigma_{cl=}$ Stresses induced in larger punch.

In both the cases, the induced compressive strength is less than the allowable yield stress of the punch material H 13 die steel i.e. (1000 MPa). So design of punch is safe under these conditions of loading.

$\sigma_y > \sigma_{cl}$ or σ_{cs}

Hence, the punch design is successful and was safe under every condition of shearing steel and the punch will perform its intended function.

Calculation of Factor of safety: Denoting by $\sigma_{w,}$ $\sigma_{y,}\sigma_u$ and n are the working stress, yield point stress and ultimate tensile stress of the material and factor of safety respectively, the magnitude of factor of safety was determined by equation. The yield point or ultimate strength of material was taken for calculating the factor of safety. The working stress of the material was taken as 337.5 MPa. **[6]**

n= σ_y/σ_w (2)

n= $\frac{1000}{335.57}$ = 2.9

Clearance allowance for Die

The clearance between the die and punch [7] can be determined as c = 0.003 Txτ.

Where

T is the sheet thickness and τ is the shear strength of sheet material.

According to distortion- energy theory, the shear stress is calculated by [8]

$\tau = 0.577 \times \sigma_{y/n}$

$\tau = 42.24$ MPa

c =0.003x19x42.24= 2.4 mm.

Case II: Calculation of load when no punch is idle

In this case, the punching starts first from smaller die as usual but the difference lies in the fact, that when the work piece is clamped to the larger die, simultaneously new work piece is clamped to the first station. That is why continuous operation was carried out. In this case, none of the station is kept idle. The rate of production of the shell of discus is high as compared to the case I. Also the power consumption is more. The load on each punch and the corresponding induced stresses are calculated as under.

$P = P_s + P_l + P_c$ (3)

$P = P_l + P_s$ (4)

P_c is neglected, as this punch will not take part in cutting operation

$P = \frac{P_s \times L}{E_s \times A_s} + \frac{P_l \times L}{E_l \times A_l}$ [9] (5)

Solving Equation (4.4) and (4.6), the load on smaller punch;

$P_s = \frac{E_s \times A_s}{E_s \times A_s + E_l \times A_l} \times P$ (6)

and the load on larger punch;

$P_l = \frac{E_l \times A_l}{E_s \times A_s + E_l \times A_l} \times P$ (7)

Taking the material of the punch holder as H13 tool steel, the modulus of elasticity is

$E_l = E_s = 215$ GPa = 215 x 10^6 MPa

A_s= Cross sectional area of smaller punch and A_l= Cross sectional area of larger punch

$A_s = \pi/4 \times 0.154^2 = 0.0186$ m^2

$A_l = \pi/4\ (D_b^2 - D_s^2) = 13.42 \times 10^3$ m^2

Taking P= 12.5 MN and after putting all the values in equation (6), load on smaller punch is

$P_s = (215 \times 10^6 \times 0.0186 / 215 \times 10^6 \times 0.0186 + 215 \times 10^6 \times 0.0320) \times 12.5 \times 10^6$

$Ps = (4 \times 10^6 / 10.5 \times 10^6) \times 12.5 \times 10^6 = 4.76$ MN

Similarly, the load on the larger punch is.

$P_l = \frac{E_l \times A_l}{E_s \times A_s + E_l \times A_l} \times P$ (8)

$P_l = (215 \times 10^6 \times 0.0302 / 215 \times 10^6 \times 0.0302 + 215 \times 10^6 \times 0.0186) \times 12.5 \times 10^6$

P_l= 6. 93/10.493) x 12.5 x 10^6 = 7.734 MN

$P = P_l + P_s$

P= 4.76 + 7.734

P = 12.5 MN

Stresses Induced in smaller and bigger punch

It was crucial here to take into account what types of stresses are induced in the punches. As the punches moves in the downward direction, it touches the sheet which is to be cut for producing the shell. If the load applied is on the top of punch holder and it is supposed that the punches are stopped by the sheet for a while, the stress induced is compressive stress. The compressive stress induced in the punches are

The stress induced on the smaller punch will be almost same as the load on punch

$\sigma_{cs} = \frac{P_s}{A_s} = 4.76 \times 10^6 / 0.0186 = 255.91$ MPa

$\sigma_{cl} = \frac{Pl}{\frac{\pi}{4} \times (D_l^2 - D_s^2)} = 7.734 \times 10^6 / 13.42 \times 10^3 = 576$ MPa

3.3 Finite Element Analysis

As main parts in the assembly were dies and punches, so FEM analysis was made for that part. Also simulation was carried out of these parts. The dies and punches were conventionally designed and later stress analysis was done by using ANSYS 14.0 software.

Mild steel C – 20 was taken for die holder. But the die holder was manufactured by H 13 steel in actual practice. Material was changed for convenience in calculating the induced stress as the shearing of mild steel offer maximum stress induced in die holder (H 13 steel).The maximum stress on H 13 steel was by shearing of mild steel C-20 steel.

3.3.1 FEM Analysis for Smaller Punch

FEM analysis of the punch holder was done in static structural module of ANSYS 14.0 in workbench. The load was applied on the plate of magnitude 4.5 MN and the smaller punch was fixed while the other two punches are kept free, according to the case I. The normal stress was found out in this analysis. Fig 5 shows the stress distribution figure of the punch holder. The various colors combination shows the stress distribution. The red color shows the area of the highest value of the stress and the blue shows the value of the lowest stress. The maximum stress found from this analysis was 278.5 MPa which was less than the permissible stress. Therefore, the design is on the safer side.

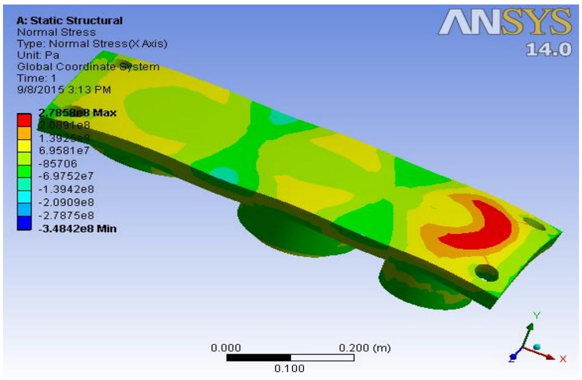

Fig 5: Stress Distribution (Smaller Punch Fixed)

3.3.2 FEM Analysis for Bigger Punch

The load applied in this analysis was 4.5 MN and the fixed support was given to bigger punch. Fig 6 shows the stress distributed view of the punch holder. The various color combination shows the stress distributions. The red colour on the extreme left shows the region of highest stress. The maximum stress found on this area was 278.5 MPa

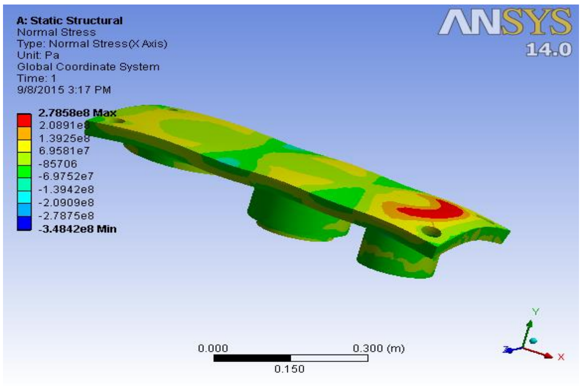

Fig 6: Stress Distribution (Bigger Punch Fixed)

3.3.3 FEM Analysis for Cavity Punch

Fig. 7 shows the stress distribution of punch holder when the cavity punch was fixed and load of 4.5 MN was applied. The areas on punch holder of smaller punch show region of highest stress.

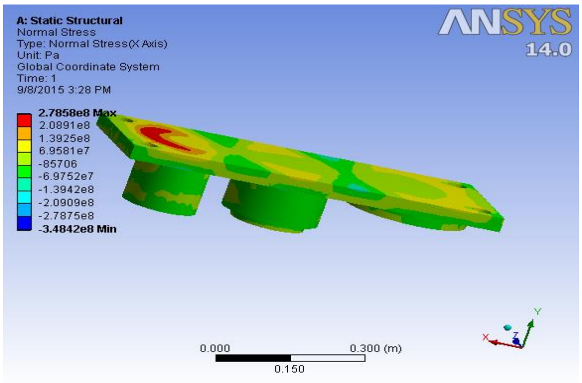

Fig 7: Stress Distribution (Cavity Punch Fixed)

3.3.4 FEM Analysis when all the Punches are fixed

In the second case, all the punches were fixed and a load of 12.5 MN was applied. The color combinations was as usual, the red color shows the area of highest stress while the blue shows the lowest value. The highest value of stress obtained was 278.5 MPa and lowest value was 69.5 MPa. Fig 8 shows the stress distribution.

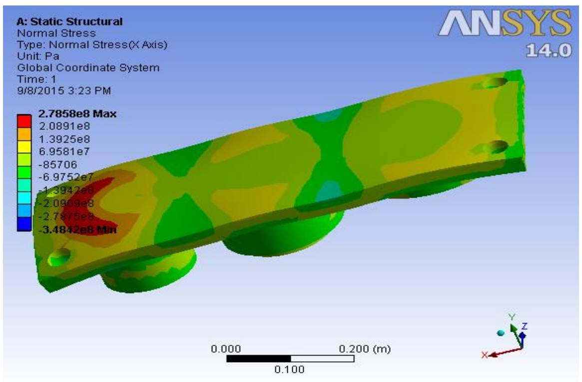

Fig 8: Stress Distribution (all punches fixed)

4. CONCLUSION

- After studying the entire manufacturing processes, and it was found that shell was not concentric. Also there was a gap remains in between both the ends. In order to join both these ends and to ensure the concentricity of the shell, welding operation and hand forging were carried out. Around 2 – 3% of the total shells produced were rejected
- Conventional Design of die assembly was done. It was found that the stresses induced in the smaller and bigger punch are 241.59 MPa and 335.57 MPa respectively. When no punch is idle, stress induced was 576 MPa. Also the induced stress is less than the permissible limit of stress of H13 steel (i.e. 1000 MPa.
- Progressive die of three stations has been developed using CATIA V5 software and parts were assembled. The punching and blanking operations are performed on first two stations and third punch was employed to give final shape to the shell
- Stress verification was done by finite element method using ANSYS 14.0 release software. The punch holder was imported from CATIA V5 software and the normal stresses were calculated. The highest stress obtained was 278 MPa which was under safe limit of allowable stress.
- Several major and minor manufacturing processes involved in shell manufacturing were combined in two operations i.e. punching and blanking.

REFERENCES

[1] Hwang, Beom Cheol, Seung Moo Han, Won Byong Bae, and Chul Kim. "Development of an automated progressive design system with multiple processes (piercing, bending, and deep drawing) for manufacturing products." The International Journal of Advanced Manufacturing Technology Vol .43 (2009) pp. 644-653

[2] Singh U. P., A. H. Streppel, and H. J. J. Kals. "Design study of the geometry of a Punching/blanking tool." Journal of materials processing technology Vol. 33.4 (1992) pp. 331-345.

[3] Jia, Zhi-Xin, Hong-Lin Li, Xue-Chang Zhang, Ji-Qiang Li, and Bo-Jie Chen. "Computer-aided structural design of punches and dies for progressive die based on functional component." The International Journal of Advanced Manufacturing Technology Vol.54 (2011) pp. 837-852.

[4] Fang, Gang, Pan Zeng, and Lulian Lou. "Finite element simulation of the effect of clearance on the forming quality in the blanking process." Journal of Materials Processing Technology Vol. 122 (2002) pp. 249-254.

[5] Al-Momani, Emad, and Ibrahim Rawabdeh. "An Application of Finite Element Method and Design of Experiments in the Optimization of sheet metal Blanking process." Jordan Journal of Mechanical and Industrial Engineering" Vol 2 (2008) pp. 53-63.

[6] Maiti, S. K., A. A. Ambekar, U. P. Singh, P. P. Date, and K. Narasimhan. "Assessment of influence of some process parameters on sheet metal blanking." Journal of Materials Processing Technology Vol.102 (2000) pp. 249-256.

[7] Klingenberg W. & Singh, U. P. "Finite element simulation of the punching/blanking process using in-process characterwasation of mild steel. Journal of Materials Processing Technology", Vol. 134(3) (2003) pp. 296-302.

[8] Hambli Ridha. "Finite element simulation of fine blanking processes using a pressure-dependent damage model." Journal of materials processing technology Vol. 116.2 (2001) pp. 252-264.

[9] Aljibori Hakim S. Sultan and Abdel Magid Hamouda. "Finite element analyswas of sheet metal forming process." Volume 33 (2009) pp. 57-69.

[10] Altan, Taylan, Nitin Jain, Xiaoxiang Shi, Gracious Ngaile, Bryan Pax, Brent Harman, and Greg Homan. "Progressive die sequence design for deep drawing round cups using finite element analyswas." Journal of manufacturing science and engineering Vol 128 (2006) 366-369.

[11] Faura F., A. Garcıa, and M. Estrems. "Finite element analyswas of optimum clearance in the blanking process." Journal of Materials Processing Technology Vol. 80 (1998) pp. 121-125.

[12] Kwak, T. S., Y. J. Kim, and W. B. Bae. "Finite element analyswas on the effect of die clearance on shear planes in fine blanking." Journal of Materials Processing Technology Vol.130 (2002) pp. 462-468.

[13] Hambli, Ridha. "Finite element model fracture prediction during sheet-metal blanking processes." Engineering Fracture Mechanics Vol. 68 (2000) pp. 365-378.