International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 10, October- 2014

Plastic Wrinkling Investigation of Sheet Metal Product Made by Deep Forming Process: A FEM Study

Mukesh Kumar
Production Engineering
Sri Balaji College of Engineering and Technolgy
Jaipur, India

Ashok Choudhary
Mechanical Engineering
Sri Balaji College of Engineering and Technolgy
Jaipur, India

Abstract- **The main objective of the study was to examine wrinkling behavior of sheet metal blank during forming of cylindrical cup. It was found that small scale industries working in this field have less technical knowledge for forming defects. The finite element method was used as a tool for this study to investigate wrinkling effect on cylindrical cup made from forming process. Experimental validation was also carried out to prove FEM simulation results. Experimental validation was based on previous research work. In current study flange wrinkling effect was study for cylindrical cup at different blank diameters ranges from 60 mm to 120 mm with two different thicknesses of 0.75 mm and 1.00 mm. An additional study was also conduct to show effect of punch velocity on product quality and wrinkling. Hoop strain and radial strain was used to show effect of wrinkling because of different conditions of forming process. FEM data show one major result that by increasing blank holder force flange wrinkling can reduce. It was also concluded that punch velocity was also responsible for flange wrinkling effect from product.**

Keywords: Sheet metal forming process, FEM, Wrinkling defects, Blank holder force

I. INTRODUCTION

Wrinkling is one of the main failure modes in deep drawing of sheet metals. In recent years, due to great usage of thin high strength sheet metals, this failure mode is becoming more prevalent in stamping of various auto motive parts. So, main emphasis has given on prediction, prevention and evaluation of wrinkling especially in final sheet metal parts. Wrinkling is a kind of local buckling of sheet metal which is formed by excessive compressive stresses. In other words, it is observed from instability under compressive stresses.

Necking, rupture, wrinkling and excessive spring back may occur in sheet metal forming processes as a failure result. A recent trend has seen in the car industry after use very thin high strength steel sheets that create defects like folding and wrinkling more often during stamping. The only feasible method capable of predicting such defects, before the real stamping operation takes place, is the Finite element method (FEM).

FEM simulation of industrial stamping processes requires in-depth knowledge of the stamping technology and a good understanding of the FEM background that is rarely addressed by companies offering commercial software. Frequently, the lack of experience in one of these two fields leads to unexpected results. This is especially important in the case of simulation of wrinkling phenomena which are closely related to sophisticated problems of instability.

II. LITERATURE REVIEW

M.R. Morovvati, B.Mollaei-Dariani et al. [1] investigated the wrinkling of two-layer (aluminum-stainless steel) sheets in the deep drawing process, through an analytical method, numerical simulations, and experiments. They found some results like: Decrease in holding length of sheet between the blank and blank holder increases the minimum required BHF to prevent wrinkling, and increase in punch diameter or blank diameter can decrease the BHF. J.P. De Magalhaes Correia, G. Ferron et al. [2] carried out an analysis of the onset of wrinkling, which was first developed for a doubly curved, elastic-plastic shell element submitted to a biaxial plane stress loading. Plastic yielding is described using a criterion recently proposed for anisotropic sheet metals. They compared the wrinkling limit curves obtained with this analysis with previous results based on different yield criteria. They also performed the Finite Element (FE) simulations of a deep-drawing experiment using the Abaqus/Explicit code with the aim of comparing the FE results relating to the initiation of wrinkling with the predictions of the analytical model and with experiments from the literature. M. Kawka, L. Olejnik et al. [3] simulated the wrinkling of conical cups using the finite element method (FEM) and verified experimentally. Two different FEM codes: static-explicit ITAS3D and dynamic-explicit ABAQUS/Explicit were used in numerical simulations. C. Loganathan, R. Narayanasamy et al. [4] carried out the experiments to find out the wrinkle of different materials having thickness 2.00 mm, for the deep drawing process using a conical die. Materials were used as ISS 19000, ISS 19600 and ISS 19660. They used the flat bottom punch for said materials of different diameters of circular blanks. Analysis of the data found by experiments shows that onset of wrinkling appears when ratio of plastic strain increments reached a critical value. Mohammad Amin Shafaat, Mahmoud Abbasi et al. [5] carried out the sidewall wrinkling of an IF-galvanized steel sheet was both experimentally and analytically examined during a conical cup wrinkling test. They used the energy method with a newly developed deflection function, to predict the critical values of stress and cup height at the onset of wrinkles, Hosford and Hill-1948 yield criteria were used. According to the obtained results, applying both the criteria results found very closer to the experimental value. They also found that newly developed deflection function has great suitability while studying about the real status of wrinkling.

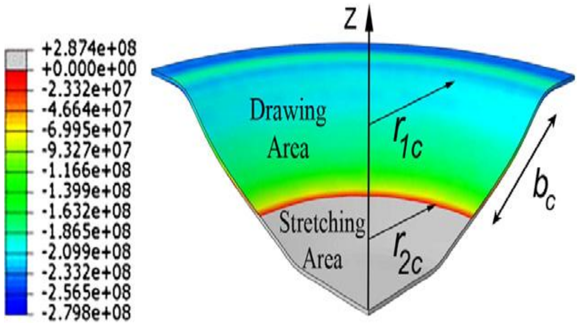


Fig. 1 Distribution of hoop stress in the deep drawn [5]

III. NUMERICAL SIMULATION

A. Introduction

Finite element method was developed because of intensive need in mechanical and civil engineering field to develop new techniques. Alexander Hrennikoff (1941) and Richard Courant (1942) were two main scientists who work to develop FEM technique in large scale. These two scientist focus on domain discretization to generate more accurate results from FEM techniques and these discretized domain formally known as elements or cells. In FEM technique Discretization is very common word and its meaning is “divide domain in continuous finite cells known as elements and have two major dividations first one was structural elements and second was unstructured element and interconnected common points at element and nodes (nodes or nodal points) and/or boundary lines and/or boundary surfaces.”

B. Theoretical Analysis Of Wrinkling

To study theory of wrinkling plasticity was most important factor and that’s why teory of plasticity was show in this section. Two stresses hoop stress and radial stress were plain stress of any cylindrical forming product was responsible for wrinkling effect. After applying punch velocity to blank sheet, the increments of strains of blank was expressed with Levy-Mises rule and show in equation form

$$d\varepsilon_1 = d\lambda[(\sigma_1 + \sigma_2)^{w-1} + (1 + 2R)(\sigma_1 - \sigma_2)^{w-1}]$$

$$d\varepsilon_2 = d\lambda[(\sigma_1 + \sigma_2)^{w-1} - (1 + 2R)(\sigma_1 - \sigma_2)^{w-1}]$$

$$d\varepsilon_3 = -d\lambda[2(\sigma_1 + \sigma_2)^{w-1}]$$

w, is assumed with of blank of sheet of cylindrical cup. Wrinkling can only occur if hoop strain is compressive in nature. There are two ratios of hoop strain and radial strains are used for final wrinkling study and show in equations form

$$\alpha = \frac{\sigma_2}{\sigma_1} = \frac{\sigma_\theta}{\sigma_r}$$

$$\beta = \frac{d\varepsilon_2}{d\varepsilon_1} = \frac{d\varepsilon_\theta}{d\varepsilon_r}$$

The quantities alpha(α) and beta(β) are the ratios of the in plane stress and the in plane plastic strain increments. Hence if the ratio of the stress is known the ration of the strain can be determined easily.

σ_r is radial stress

σ_θ is hoop stress

w is wealth of blank sheet.

III. EXPERIMENTAL VALIDATION AND DESIGN OF EXPERIMENT

A. Introduction

Finite element method is most useful technique in research field, because it is time and money saving approach, but it has also some issues related to its validation with real environment. In this thesis it was also take care to validate FEM simulation with real environment. There are two approaches to validate any FEM simulation work, first is setup new experimental test module and perform test on it and validate it with FEM simulation, but it want time and money investment. Second approach is to use previous published experimental work for FEM simulation validation. In this study second approach was used.

B. Steps Involved In Validation Process Are Following

Step I Research Paper selection: For this study, Research paper selection was based on SCI index report. Title of research paper is “Wrinkling behavior of laminated steel sheets, Journal of Materials Processing Technology, 2004”[6]

Step II Geometry Generation: in this step same geometry was generated to exact validation with simulation work.

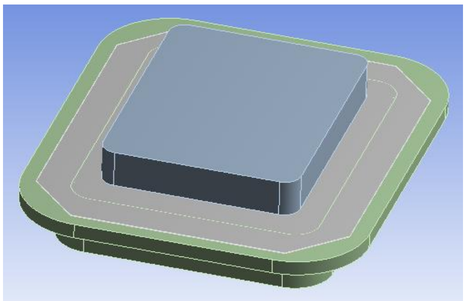


Fig. 2 CAD design of Punch, blank and Die used for Validation (PP 138)

Step III Material Selection: For validation it was most important to use same material as used in previous work.

Step IV Boundary Conditions: Same boundary conditions were used in this section to get exact validation with simulated work

Step V Result Comparison: this step was last step in experimental validation process. There is no need to validate all results with experimental results. Only some results can verify simulation results. According to this paper used for experimental simulation if binding force limit set to 80 KN limit for solid 1 then experimental product wrinkling (pp 139, fig 19) and simulation product wrinkling was shown in fig.3 and show similar results. When wrinkling at corner was measure at simulated case it was approx 0.7 mm and in experimental work it was at 0.4 mm (pp 139, table 3).

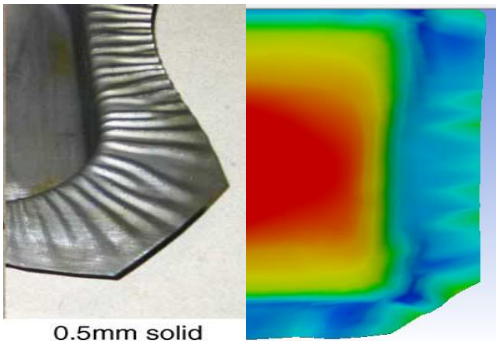


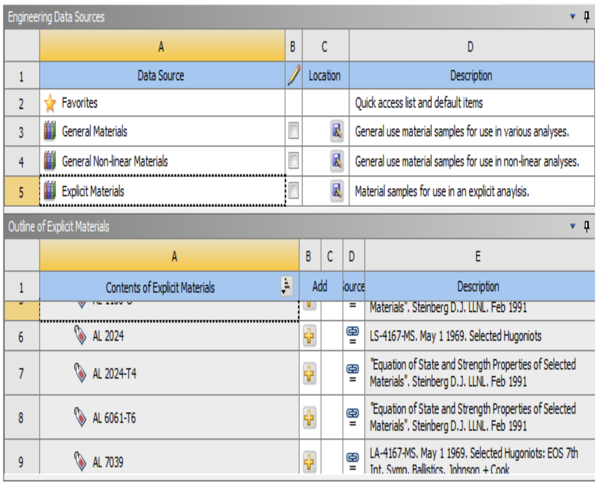
Fig. 3 Experimental vs simulation wrinkling effect

C. Steps Covered In Fem Simulation

ANSYS EXPLICT DYNAMICS is highly developed FEM code and used for various type of problems. Every feature in this code is described by top to bottom steps which were shown in following part of this thesis. Various material models are used by this code according to problem statement.

Pictorial presentation of steps followed in this code.

Step I. Engineering Data: In this step material properties were defined in software using engineering data panel. Here various predefined material models were given which are used according to problem statements. Fig. 4 shows the engineering data panel used in this study.



Engineering Data Sources

	A	B	C	D
1	Data Source		Location	Description
2	Favorites			Quick access list and default items
3	General Materials			General use material samples for use in various analyses.
4	General Non-linear Materials			General use material samples for use in non-linear analyses.
5	Explicit Materials			Material samples for use in an explicit anaylsis.

Outline of Explicit Materials

	A	B	C	D	E
1	Contents of Explicit Materials	Add		ource	Description
					Materials". Steinberg D.J. LLNL. Feb 1991
6	AL 2024				LS-4167-MS. May 1 1969. Selected Hugoniots
7	AL 2024-T4				"Equation of State and Strength Properties of Selected Materials". Steinberg D.J. LLNL. Feb 1991
8	AL 6061-T6				"Equation of State and Strength Properties of Selected Materials". Steinberg D.J. LLNL. Feb 1991
9	AL 7039				LA-4167-MS. May 1 1969. Selected Hugoniots: EOS 7th Int. Symp. Ballistics. Johnson + Cook

Fig. 4 Engineering Data panel in Ansys Explicit Dynamics Code

Step II. Geometry making: Geometry making is essential part of FEM modeling. In this study geometry making were done by CAD software Autodesk Inventor (version) and imported to this code in WB (work bench environment). In Fig.5 CAD design is shown. Healing of design was completed in Ansys software.

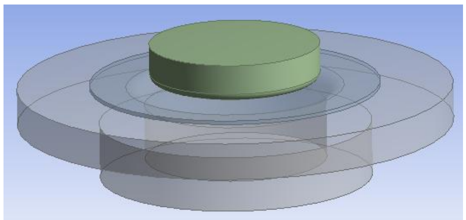
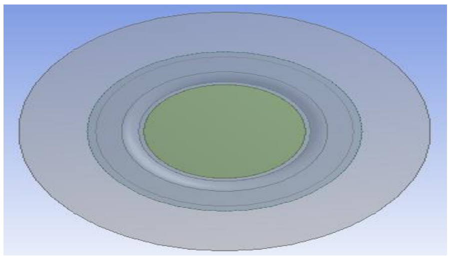


Fig. 5 Isometric view of Test Plate and Tool



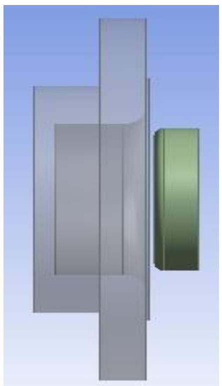


Fig. 6 Front & side view of Test Plate and Tool

Step III. Meshing of Domain: As described in earlier sections meshing is minimum requirement to perform simulation using this code. Automatic mesh generation tool was used in this study. Fig.7 shows the GUI image of this step.

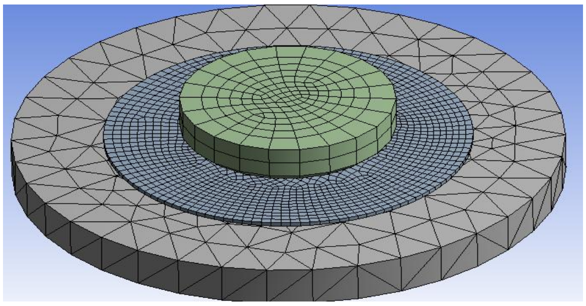


Fig. 7 GUI image of Meshed Domain

Step IV. Boundary Conditions and assumptions: The following assumptions are used in current study.

- Impact tool is considered as rigid in analysis.
- Initial velocity of tool is in negative z direction only.
- Automatic mesh sizing tool is used for mesh generation.
- General material model is used for tool material
- Time step is selected by assumption only.

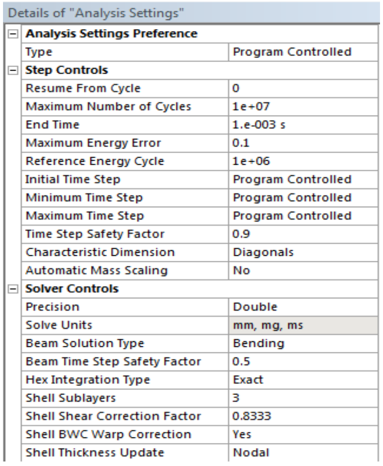
In this study test plate was considered as fixed support plate and tool is considered as bullet with some initial velocity and was shown in fig.8. Here initial velocity at time zero is also zero. In FEM simulation part, a range of velocity was used for analysis.

Details of "Punch"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Rigid
Reference Temperature	By Environment
Reference Frame	Lagrangian
Material	
Assignment	Structural Steel
Bounding Box	
Properties	
Statistics	

Fig. 8 Boundary Conditions in Code

Step V. Analysis Settings: In this step FEM analysis time and space setting were given to present problem. End time was set to 3E-03 sec and initial time was set to program controlled. Other properties were shown in fig.9.



Details of "Analysis Settings"

Analysis Settings Preference	
Type	Program Controlled
Step Controls	
Resume From Cycle	0
Maximum Number of Cycles	1e+07
End Time	1.e-003 s
Maximum Energy Error	0.1
Reference Energy Cycle	1e+06
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled
Maximum Time Step	Program Controlled
Time Step Safety Factor	0.9
Characteristic Dimension	Diagonals
Automatic Mass Scaling	No
Solver Controls	
Precision	Double
Solve Units	mm, mg, ms
Beam Solution Type	Bending
Beam Time Step Safety Factor	0.5
Hex Integration Type	Exact
Shell Sublayers	3
Shell Shear Correction Factor	0.8333
Shell BWC Warp Correction	Yes
Shell Thickness Update	Nodal

Fig. 9 Analysis Setting in Ansys

Step VI. Post Processing (Results)

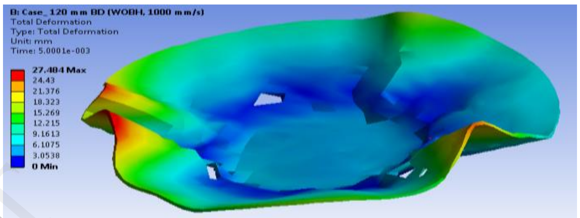


Fig. 10 Total deformation of metallic sheet

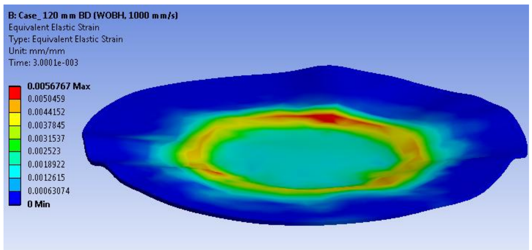


Fig. 11 Elastic strain of metallic sheet

D. Material Properties Of Plate And Tool

In this study three materials were used for analysis from them two were composite materials and one is alloy metal. In Table 1 & 2 material properties of all materials are shown. Structural steel was used for projectile tool.

TABLE 1 PROPERTIES OF STRUCTURAL STEEL

Property	Value	Unit
Density	7850	Kg/m^3
Young's Modulus	2E+11	Pa
Poisson's Ratio	0.3	
Bulk Modulus	1.66E+11	Pa
Shear Modulus	7.692E+10	Pa
Sp. Heat	434	J/kg C

TABLE 2 PROPERTIES OF SS 304

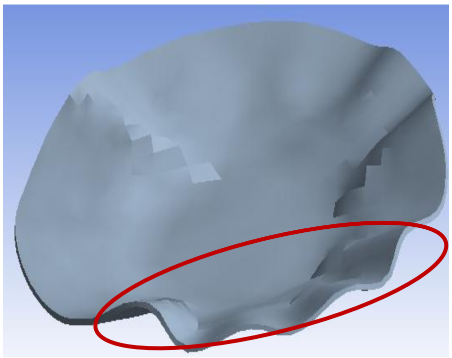
Property	**Value**	**Unit**
Density	7900	Kg/m^3
Sp. Heat	423	J/kgC
Steinberg Guinean Strength		
Initial Yield Stress	3.4E+08	Pa
Max Yield Stress	2.5E+08	Pa
Hardening Constant	43	NA
Hardening Exponents	0.35	NA
Derivative dG/dP	1.74	NA
Derivative dG/dT	-3.504E+07	Pa/C
Derivative dY/dP	0.007684	NA
Melting Temperature	2106.9	C
Shear Modulus	7.7E+10	Pa
Shock EOS Linear		
Gruneisen Coefficient	1.93	NA
Parameter C1	4570	m/s
Parameter S1	1.49	NA
Parameter Quadratic S2	0	s/m

In this study orthographic material properties were used so that FEM results are related more perfectly with real world environment. Tool material is only elastic material and this problem can resolve by converting it to rigid body.

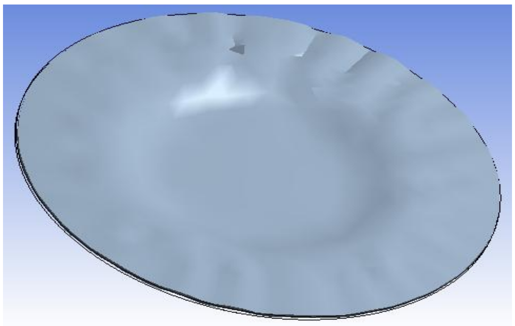
IV. RESULT AND DISCUSSION

In this study various process parameters were simulated to find out wrinkling effect like application of blank holder force or binder force, punch velocity, thickness of blank, initial blank diameter and many more. In this study wave number also calculated to show wrinkling effect for all cases. Wrinkling effect was very common problem in draw forming process and there are many methods to reduce it and very common was use of blank holder with certain force limit. This can reduce wrinkling effect from final product. Another important issue about wrinkling is material thickness. SS 304 with different thickness were used in FEM simulations.

Flange wrinkling is very common problem in sheet metal form based products and reason behind this is undesirable deformation of sheet, thickness and compressive stresses. When drawing process starts and increase draw depth after certain height, the flange region plastically buckles into a number of waves and the process fails down. Wrinkling effect can be reduced by application of blank holder with certain force limit which was shown in fig. 12 as a sample.



(A) Without blank holder



(B) With blank holder

Fig. 12 With and without blank holder draws formation having wrinkling effect

Another way to understand wrinkling effect was linear graph generation between hoop strain and radial strain.

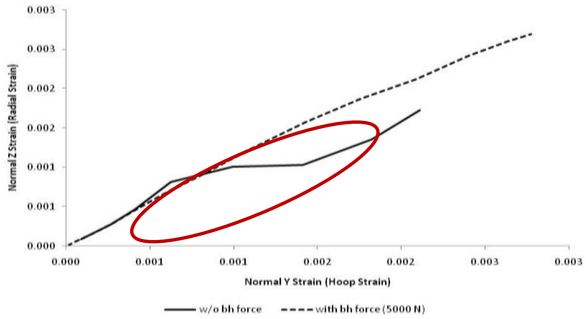


Fig. 13 Hoop and radial strain for 100 mm blank of 1 mm thickness

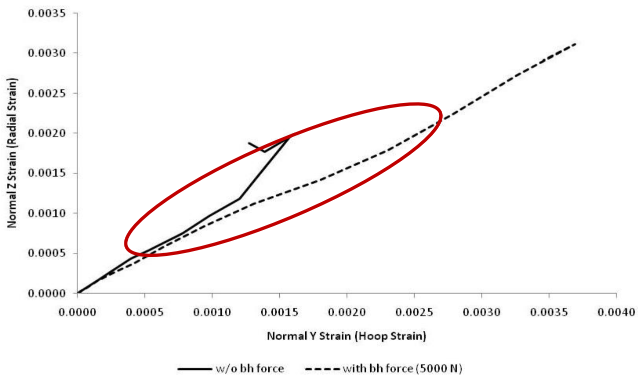


Fig. 14 Hoop and radial strain for 100 mm blank of 0.75 mm thickness

It was observed that when hoop strain and radial strain was maintain linear relationship no wrinkling was occur in drawing operation but when sudden change in linear line occur (red zone in this section see fig. 13 and 14) means wrinkling occur.

V. CONCLUSION

In this study the commercial software ANSYS WB software was used for drawing formation simulation to know about wrinkling effect. The effect of blank holder force on flange wrinkling is analysis for drawing quality at SS 304 material. Hoop and radial strain diagrams are shown to show that blank force can reduce wrinkling effect.

Main conclusions were following in next section.

- Blank holder forces from 0 N to 7000 N are simulated from 1 mm and 0.75 mm SS304 steel material. For 0 N blank holder force, 9 waves are observed on the flange region of plate. After performing simulations with blank holder force, it was observed that in order to prevent wrinkling 5000 N blank force was sufficient.
- The blank holder force analysis show that as the blank holder force increases for certain limit, the wave number increases and the wave amplitude decrease see figure from previous chapters and appendix data.
- Effect of blank diameter was also important parameter for wrinkling effect; it was observed that as blank diameter was increased amplitude of wrinkling was also increased.

VI. FUTURE SCOPE

Although lots of work is doing by worldwide by researchers but this field is growing field and now-a-days FEM simulation is necessity for any production industry.

In this study all main points was focused but some points was neglected which can be improved in future like:-

- Computational time limit draw for 3 to 5 mm in future try to draw for 20 mm to 50 mm
- Mesh quality improvement
- Various different shapes like tapered shape etc.
- Wall wrinkling effect.

REFERENCES

[1] M.R. Morovvati, B.Mollaei-Dariani, M.H.Asadian-Ardakani, “A theoretical, numerical, and experimental investigation of plastic wrinkling of circular two-layer sheet metal in the deep drawing” Journal of Materials Processing Technology vol-210 pp. 1738–1747, 2010

[2] J.P. De Magalhaes Correia, G. Ferron, L.P. Moreira, “Analytical and numerical investigation of wrinkling for deep-drawn anisotropic metal sheets” International Journal of Mechanical Sciences vol-45 pp. 1167-1180, 2003

[3] M. Kawka, L. Olejnik, A. Rosochowski, H. Sunaga, A. Makinouchi, “Simulation of wrinkling in sheet metal forming” Journal of Materials Processing Technology vol-109 pp. 283-289, 2001

[4] C. Loganathan, R. Narayanasamy, S. Sathiyanarayanan, “Effect of annealing on the wrinkling behaviour of the commercial pure aluminium grades when drawn through a conical die” Materials and Design vol-27 pp. 1163–1168, 2006

[5] Mohammad Amin Shafaat, Mahmoud Abbasi, Mostafa Ketabchi, “Effect of different yield criteria on prediction of wrinkling initiation of interstitial-free galvanized steel sheet” Materials and Design vol-32 pp. 3370-3376, 2011

[6] H.S. Cheng, J. Cao, H. Yao, S.D. Liu, B. Kinsey, “Wrinkling behavior of laminated steel sheets” Journal of Materials Processing