All Sciences Proceedings
http://as-proceeding.com/

1st International Conference on Pioneer and Innovative Studies
June 5-7, 2023 : Konya, Turkey

ICPIS
https://as-proceeding.com/index.php/icpis

Effect Of Radius On Deep Drawing Stress Analysis

Recep Özbek[1], Okan Gül[*2]

[1]Mechanical Engineering Department, Kocaeli University, Türkiye
[2]Mechanical Engineering Department, Kocaeli University, Türkiye

[*](okan.gul@kocaeli.edu.tr)

Abstract – In this study, die, punch and sheet metal are designed in Solidworks package program, and then radius are given to die and punch, and stress analyzes are examined in Ansys package program. The results of the analysis resulted as we expected, and as the amount of radius we gave increased, the stresses decreased. The aim of this study is to show the effect of radius in the deep drawing process and to show that both time saving and economy can be achieved thanks to analysis programs such as Ansys package program.

Keywords – Deep Drawing, Radius, Punch, Sheet, Ansys Analysis

I. INTRODUCTION

The most important method used to obtain cup-shaped parts from sheet metal is deep drawing [1]. Today, with its frequent use, it has been further developed and micro-level shaping is done by micro-deep drawing process. In the deep drawing process, round sheet plates made of steel and aluminum materials are generally used. This process gives permanent shape to the sheet metal plates through permanent deformation, i.e. plastic forming. Usually, cylindrical containers are created with this method. Most of the time this process is done in successive phases in one go. The containers formed using the deep drawing method are shown in Fig. 1 [2].

Fig. 1 The vessels formed by the deep drawing method

As seen in Fig. 2, the deep drawing process generally consists of punch, die and sheet metal. In addition, "In practice, if D/d (diameter of sheet metal / diameter of punch)>21/20, pot ring is used. If D/d<21/20, deep drawing can be done without using pot ring." [1] The theoretical analysis of the deep drawing process of cylindrical containers was studied for the first time by Hessenberg [3] and Danckert [4], and the effect of the stresses generated during drawing on the die profile was investigated.

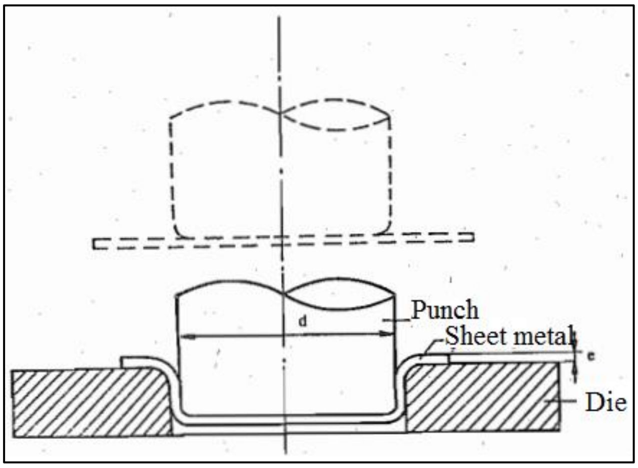


Fig. 2 Punch, sheet metal (drawing sheet), die used in deep drawing process

In 1996, Mamalis investigated the deep drawing of cylindrical containers using the express finite element technique (with the help of the DYNA 3D program) by simulating its effect on the forming characteristics of the material. Theoretical results were compared with the experimental results, depending on the elongation distribution (radial, circumferential and throughout the thickness) for 5 different galvanized sheet and aluminum sheet by varying the punch force and punch movement distance, and a good agreement was observed [5, 6].

The punch is pressed against the blank deep drawing process begins, that begins to deform. The deep drawing force is small, at the beginning of the process, therefore only slight local deformation takes place, after which, as the stress increases, the material enters the plastic deformation and begins the actual deep drawing process [7, 8, 9].

Since ANSYS student version was used in this study, deep drawing depth had to be kept small due to limitations. In addition, pot circle was not used due to the use of ANSYS analysis program. It can be known that the deep drawing technology for finally obtaining the optimal parameters for unbuffered drawing is a combination of different fields of scientific research, which are: mathematical, mechanical, computer science, etc. with software analysis and computer simulation software [10, 11, 12].

Therefore, numerical computer simulations which uses finite element analysis and laboratory simulations tend to substitute industrial testing for the purpose of implementing real-world results [13, 14, 15].

Since there is no direct mathematical formula to calculate the stresses or deformations in a body having a complex form in the finite element analysis, it is done by using simple mathematical models that can calculate these characteristics in bodies with simple geometric shapes (cube, pyramid) when certain tasks are applied to them [16, 17].

Hence, the utilization of the Finite Element Method (FEM) is based on the meshing of a complex, subject-to-demand form in regular elements for which internal stresses and deformations are able to be calculated [18].

The main reason why analysis programs are used in such studies is to provide economic benefits. Before a process is actually performed on the bench, it can be studied with analysis programs such as ANSYS, Auto-form, Simufact, and it can be learned by simulating beforehand where there will be tears and where there will be difficulties. Of course, simulation and practice do not give the same result, but due to these simulations, a lot of time and money has been saved today.

II. Materials and Method

Using Solidworks package program, 0.8 mm thick sheet material, male and female die were designed.

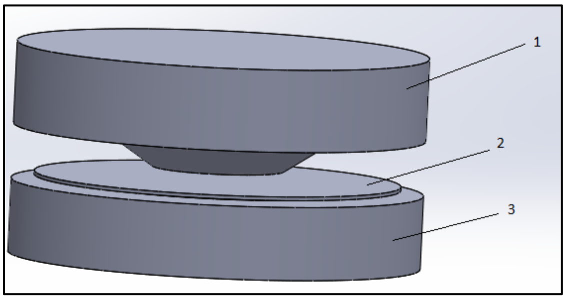


Fig. 3 Design of deep drawing assembly. 1-punch, 2-sheet metal, 3-die

These parts were designed separately in the Solidworks package program and then imported into the ANSYS package program as in Fig. 3. After importing the geometry, the model section was selected, and the male and female dies were selected as rigid, the sheet material flexible was selected. While the material of the male and female molds was chosen as classical structural steel, the material of the sheet was chosen as Aluminum Alloy NL as in Fig. 4.

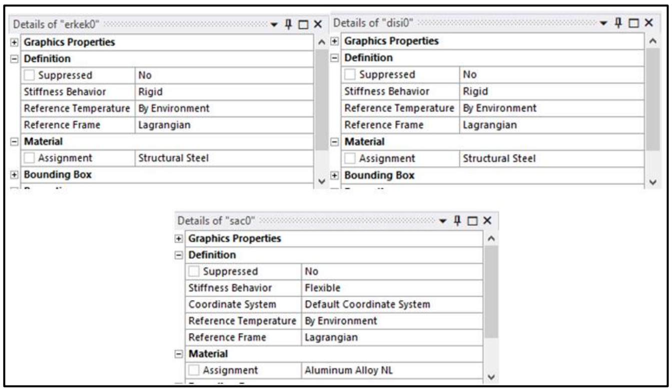


Fig. 4 Material and definition section in Ansys program

Mechanical properties of classical structural steel material and aluminum material are shown in Fig. 5 and Fig. 6, respectively. These mechanical properties were selected from the Ansys program and used to perform analysis.

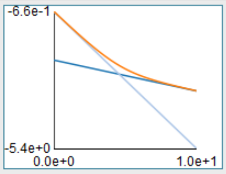
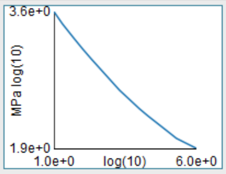
Derive from	Young's Modulus and Poisson's Ratio
Young's Modulus	2e+05 MPa
Poisson's Ratio	0,30000
Bulk Modulus	1,6667e+05 MPa
Shear Modulus	76923 MPa
Isotropic Secant Coefficient of Thermal Expansion	1,2e-05 1/°C
Compressive Ultimate Strength	0 MPa
Compressive Yield Strength	250,00 MPa
Strain-Life Parameters	
S-N Curve	
Tensile Ultimate Strength	460,00 MPa
Tensile Yield Strength	250,00 MPa

Fig. 5 Mechanical properties of classical structural steel

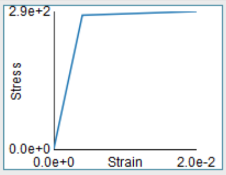
Derive from	Young's Modulus and Poisson's Ratio
Young's Modulus	71000 MPa
Poisson's Ratio	0,33000
Bulk Modulus	69608 MPa
Shear Modulus	26692 MPa
Bilinear Isotropic Hardening	

Fig. 6 Mechanical properties of Aluminum Alloy NL

III. Results and Discussion

In the contact part, two frictionless contacts are used as in Fig. 7.

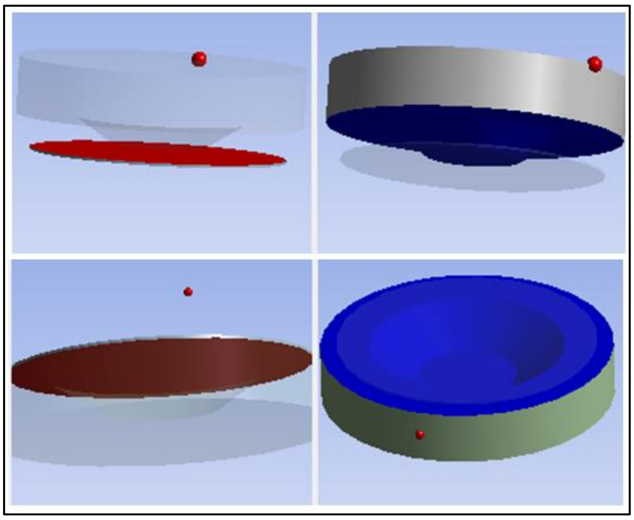


Fig. 7 Contacts between the punch and the sheet

In the mesh phase, a generate mesh was created so that the element quality value would not fall below 3, both from the limits of the Ansys student version and to shorten the simulation time, and a 3 mm mesh was added to the sheet metal body size. The analysis time was chosen as 0.001 s and fixed support was displaced from the down plate and displacement was displaced from upper plate. Radius 0-3-6-9-12 one by one and Radius in both die and punch (12-4) were given and this analysis was repeated, and the results were given. The stress results are shown as follows.

Explicit dynamics analysis results without using any radius are given in Fig 8 and Fig 9. The maximum and minimum equivalent (von-mises) stress was obtained as 465.91 MPa and 59.599 MPa, respectively. The maximum and minimum normal stress (Y-axis) was obtained as 71.844 MPa and -574.71 MPa, respectively.

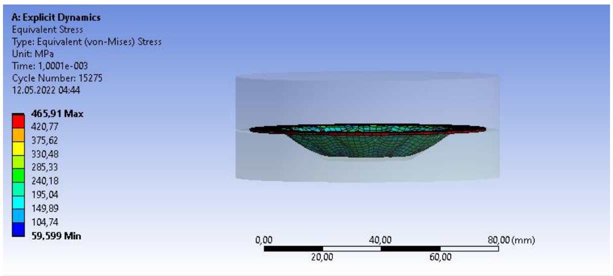


Fig. 8 Equivalent (von-mises) stress without using any radius

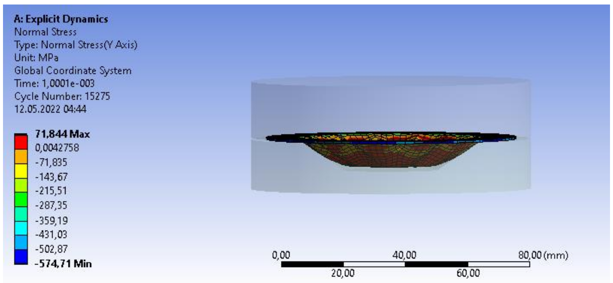


Fig. 9 Normal stress (Y-axis) without using any radius

Explicit dynamics analysis results are given in Fig 10 and Fig 11. A radius of 4 mm on the lower surface of the die was used in this and every subsequent analysis, and a radius of 3 mm was used on the surface of the die in this analysis. The maximum and minimum equivalent (von-mises) stress was obtained as 462.54 MPa and 27.324 MPa, respectively. The maximum and minimum normal stress (Y-axis) was obtained as 61.613 MPa and -605.03 MPa, respectively.

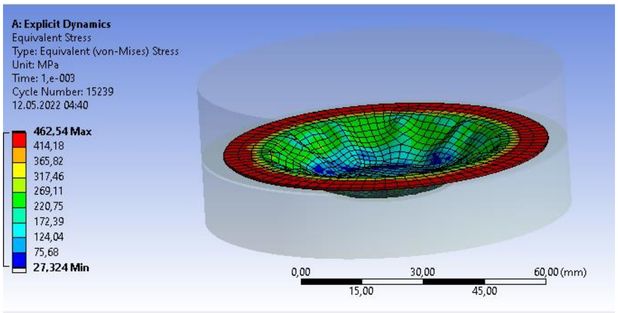


Fig. 10 Equivalent (von-mises) stress for 3 mm radius

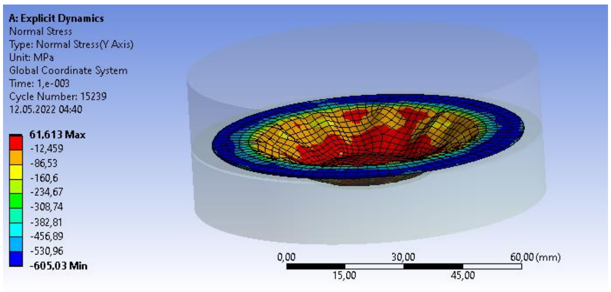


Fig. 11 Normal stress (Y-axis) for 3 mm radius

Explicit dynamics analysis results with using 6 mm radius are given in Fig 12 and Fig 13. The maximum and minimum equivalent (von-mises) stress was obtained as 466.06 MPa and 15.729 MPa, respectively. The maximum and minimum normal stress (Y-axis) was obtained as 51.418 MPa and -545.46 MPa, respectively.

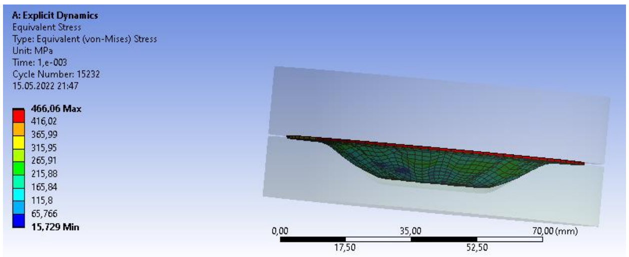


Fig. 12 Equivalent (von-mises) stress for 6 mm radius

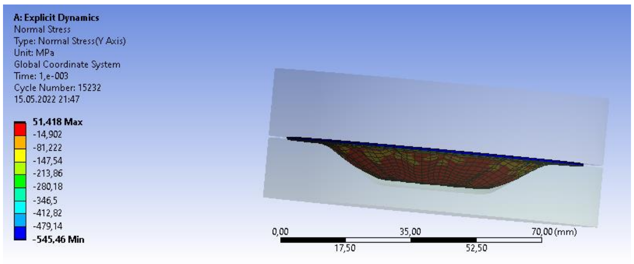


Fig. 13 Normal stress (Y-axis) for 6 mm radius

Explicit dynamics analysis results with using 9 mm radius are given in Fig 14 and Fig 15. The maximum and minimum equivalent (von-mises) stress was obtained as 499.07 MPa and 12.962 MPa, respectively. The maximum and minimum normal stress (Y-axis) was obtained as 53.625 MPa and -549.69 MPa, respectively.

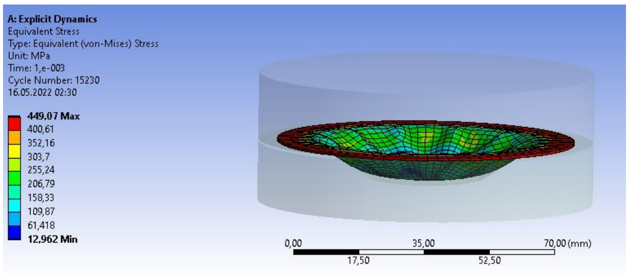


Fig. 14 Equivalent (von-mises) stress for 9 mm radius

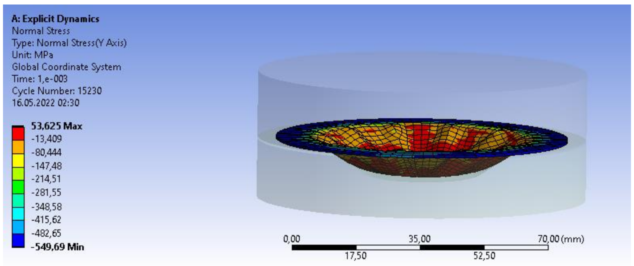


Fig. 15 Normal stress (Y-axis) for 9 mm radius

Explicit dynamics analysis results with using 12 mm radius are given in Fig 16 and Fig 17. The maximum and minimum equivalent (von-mises) stress was obtained as 451.94 MPa and 8.5188 MPa, respectively. The maximum and minimum normal stress (Y-axis) was obtained as 54.803 MPa and -539.03 MPa, respectively.

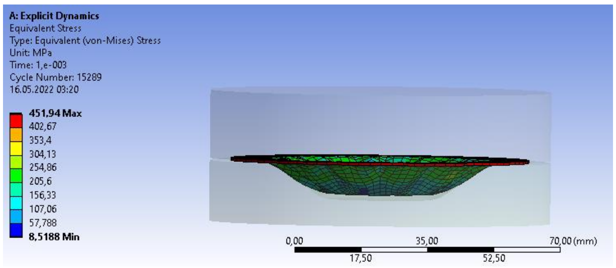


Fig. 16 Equivalent (von-mises) stress for 12 mm radius

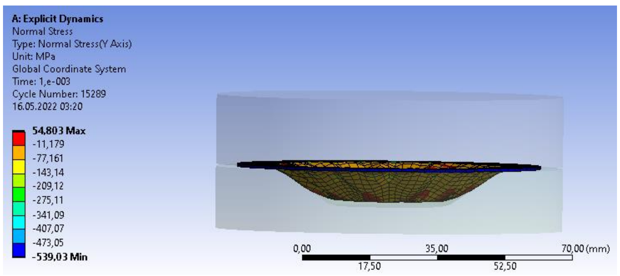


Fig. 17 Normal stress (Y-axis) for 12 mm radius

Explicit dynamics analysis results are given in Fig 18 and Fig 19. 4 mm radius on the lower surface of the female mold and 12 mm radius on the upper surface, in addition to the 4 mm radius on the punch was used. The maximum and minimum equivalent (von-mises) stress was obtained as 445.8 MPa and 10.746 MPa, respectively. The maximum and minimum normal stress (Y-axis) was obtained as 32.26 MPa and -540.66 MPa, respectively.

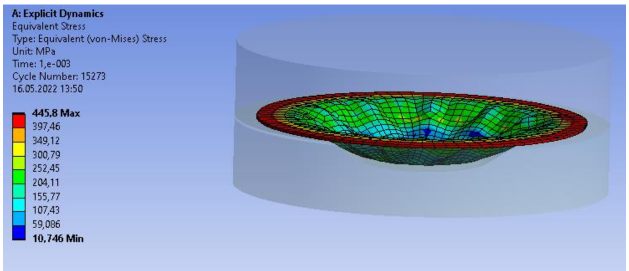


Fig. 18 Equivalent (von-mises) stress for 12 mm radius and 4 mm punch

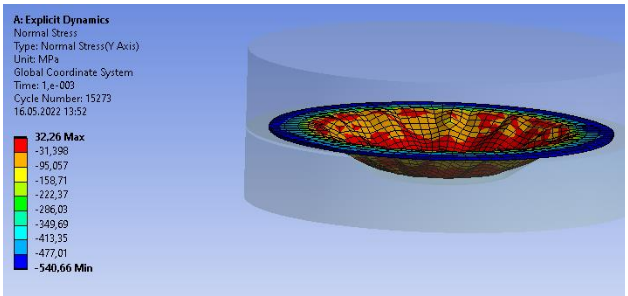


Fig. 19 Normal stress (Y-axis) for 12 mm radius and 4 mm punch

IV. CONCLUSION

This study showed that although the given radius is not always as expected, there is a significant difference between the initial stress values without radius and the stress values with radius in both the punch and the die. This is the expected result because it is aimed to reduce these stresses by giving radius to the points where stresses can occur. While the normal stress value is max 71.844 MPa, the minimum value 32.26 MPa is seen. This result shows that radius values have a significant effect on normal stress. Likewise, the equivalent stress value formed in the tool without any radius is 465.91 MPa, while the equivalent stress value is 445.8 MPa where 4 mm, 12 mm, and 4 mm radius are given to the punch, die upper and die lower surfaces, respectively. This is noticeable decline. As a result, thanks to many analysis programs such as Ansys and many more, as time passes and technology progresses, practical and simulation values will approach each other and instead of trying repeatedly for studies, simulations will be used to save both economy and time.

REFERENCES

[1] Çapan L., Metallere Plastik Şekil Verme, 7th ed., Çağlayan Kitapevi, Istanbul, 2020.

[2] https://en.wikipedia.org/wiki/Deep_drawing#/media/File:Deep_Draw_Line_example_by_Pressteck_Italy.jpg.

[3] Hessenberg W.C., A simple Account of Prof. Swift Work on Deep Drawing, BISRA Report, 1954.

[4] Danckert J., Reduction of Residual Stree in Deep Drawwing by Modelling the Draw Die profile, Ann. CIRP 44 (1) (1995) 259-262.

[5] Mamalis A.G., Manolakos D.E., Baldoukas A.K. "Simulation of Sheet Metal Forming Using Eksplisit Finite-Element Techniques: Effect of Material and Forming Characteristics Part 1. Deep-drawing of Cylindrical Cups", Journal of Materials Processing Technology 72 (1997), 48-60.

[6] Mamalis A.G., Manolakos D.E., Baldoukas A.K., "Simulation of Sheet Metal Forming using Eksplisit Finite-Element Techniques, Effect of Material and Forming Characteristics Part 2. Deep-drawing of Square Cups", Journal of Materials Processing Technology 72 (1997), 110-116.

[7] Göl S., Derin Çekme İşlemine Etki Eden Parametrelerin Analizi ve Optimizasyonu, Yüksek Lisans Tezi, Uludağ Üniversitesi, Fen Bilimleri Enstitüsü, Bursa, 2005, 168202.

[8] Krishna K.M., Swamy M. K., International Refereed Journal of Engineering and Science 3, (2014), 24-29.

[9] Choubeya A. K., Agnihotrib G., Sasikumarc C., Singhd M., Materials Today: Proceedings 4, (2017), 2511–2515.

[10] Torkar M., Tehovnik F., Podgornik B., Engineering Failure Analysis 40, (2014), 1 – 7.

[11] Dwivedia R., Agnihotrib G., Materials Today: Proceedings 4, (2017), 820 – 826.

[12] Kumar R.U., Reddy G. C., International Journal of Advanced Materials Manufacturing & Characterization 3, (2013), 381 – 384.

[13] Das P., Panda S.K., Pratihar D.K., Advanced Materials Manufacturing & Characterization 3, (2013), 99-104.

[14] J.P. Fan, C.Y. Tang, C.P. Tsuib, L.C. Chan, T.C. Lee, International Journal of Machine Tools & Manufacture 46, (2006), 1035 – 1044.

[15] M. Emad, S. Soliman, M. G. El-Sebaie, Journal of Materials Processing Technology 216, (2015), 520–528.

[16] G. Chandra Mohan Reddy, P. V. R. Ravindra Reddy, T. A. Janardhan Reddy, Tribology International 43, (2010), 1132–1137.

[17] Ravindra K. Saxena, P.M. Dixit, Finite Elements in Analysis and Design 47, (2011), 1104– 1117.

[18] R. Padmanabhana, M.C. Oliveira, J.L. Alves, L.F. Menezes, Finite Elements in Analysis and Design 43, (2007). 1062 – 1067.

[19] T. Hama, T. Hatakeyama, M. Asakawa, H. Amino, A. Makinouchi, H. Fujimoto, H. Takuda, Finite Elements in Analysis and Design 43, (2007), 234–246.