Finite Element Simulation of Deep Drawing Parameters Effects on Cup Wall Thickness

Dr. ABDUL KAREEM FLAIH HASSAN
Department of Mechanical Engineering
University of Basrah
College of Engineering

ALI HASOON ABDULHADI
Department of Mechanical Engineering
University of Basrah
College of Engineering

***Abstract-* The present research aims to predict the thickness distribution of a wall of a deep drawn cup. A simplified 3D axisymmetric model which represents the deep drawing set (blank and tools) was created using a CAD software, and then imported into a finite element code ANSYS where a simulation was carried out. The model represents a cylindrical cup made of low carbon steel sheet. The results showed that the FE model represents real deep drawing process fairly well. The cup thickness distribution values showed a good agreement with the referenced values, where the failure or success of drawing process could be predicted based on the obtained thickness results. It was observed that a high value of friction restrains material movement and resulted in producing more thinning and more punch force. High blank holder force was found to decrease the thickness of both the bottom face of the cup and the flange rim. While increasing die corner radius increases thickness and the maximum thinning occurred at the smallest die corner radius. It was found by decreasing the punch profile radius the thickness at the flat bottom of the cup and under the punch profile region were reduced.**

1. Introduction

Deep drawing is one of the extensively used sheet metal forming processes in the industries to have mass production of cup usually have complicated shapes in a very short time **[1]**. Deng and Blesi (1999)**[2]** Proposed new theoretical models for predicting the drawing fracture load of an axisymmetric cup drawing. These models take into account the influence of tri axial stress state, anisotropy, strain hardening, bending, and tool geometry. The optimum punch profile radius is found to be between 5-7 times the thickness of the sheet. Hong Yao, Jian Cao (2000)**[3]** formulated an analytical model to calculate the offset of the simplified axisymmetric model for predicting the failure height of 3D parts. Simulation results demonstrate that the corner stretch height is always larger than the side stretch height for square cup .The accuracy of the failure height prediction using the 2D model with offset has been improved. Hun and Kim (2001) **[4],** they applied FE analysis to estimate the initial blank shape, intermediate deform shape, thickness distributions and failure during multistage deep drawing operation of elliptic cup. Khalil (2006) **[5]** proposed a numerical procedure for the design of deep drawing process using FEM. A simplified 2D axisymmetric model of cylindrical cup of 43 mm outer diameter and 0.5 mm thickness drawn from mild steel blanks had been developed. The research aimed to study the effect of some parameters which influence the drawing process, such as friction coefficient, blank holding force, punch and die corner, and to predict the tearing failure in drawn parts.

2. Building of the FE model

The ultimate purpose of a finite element analysis is to recreate mathematically the behavior of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. This model comprises all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system **[6]**.

2.1 Model Generation

A global cartesian coordinate system is used and working plane is the *x-y* plane where ***z*** axis is the normal. The first step involved creating the key points, Figure.1, each point has ***x*,*y*, *z*** values defining its location in space, before creating the lines that connect the key points together, the blank diameter needed for the given cup height must be calculated. The blank size for cylindrical parts with small punch radius (*r* <10 mm) is calculated as follows [7]:

i. Cup without flange(Figure.2):

$$D = \sqrt{d^2 + 4dh}$$

ii.Cup with flange:

$$D = \sqrt{d^2 + 4dh} - 0.5R$$

$$\frac{d}{R} = 15\ to\ 20$$

$$D = \sqrt{d^2 + 4dh} - R$$

$$\frac{d}{R} = 10\ to\ 15$$

$$D = \sqrt{(d-2R)^2 + 4d(h-R) + 2\pi R(d-0.7R)}$$

$$\frac{d}{R} < 10$$

Where the blank diameter, is the bunch diameter, is the cup height, and is the die corner radius.
The following step includes joining the existing key points with lines, Figure.3, to only one quarter of the geometry is modeled taking advantage of symmetry to reduce computational expense. Next is to generate a 3D model

from these 2D entities, Figure.4.

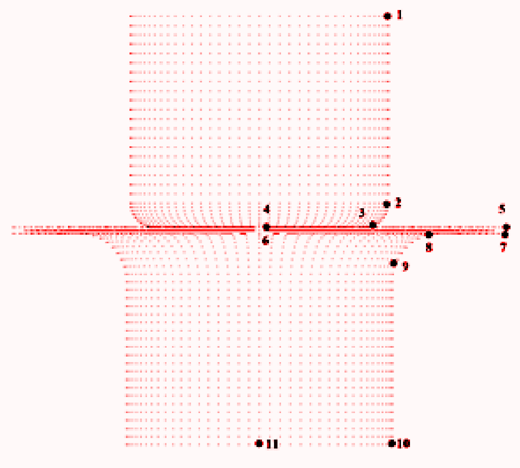

Fig.1 Key points created inside ANSYS

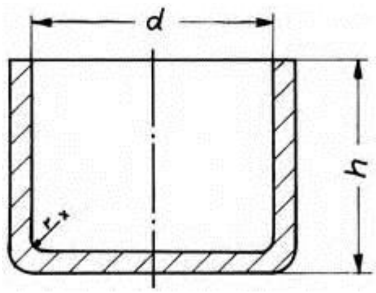

Fig.2 Cup with no flange

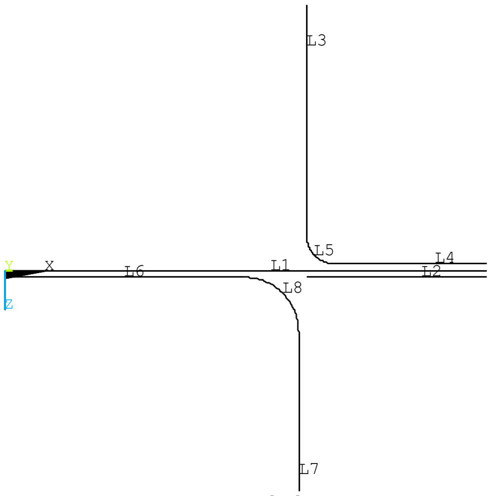

Fig.3 Lines joining the key points

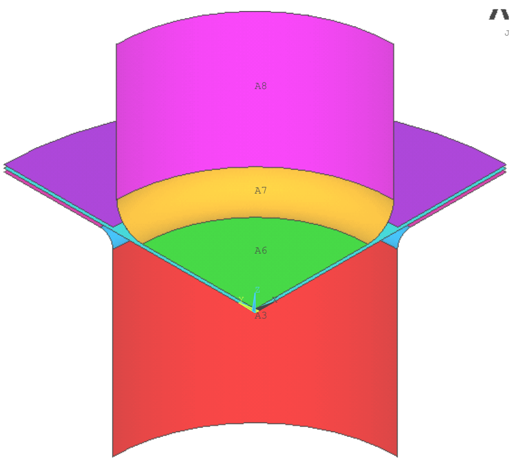

Fig.4 Areas resulting from lines rotation.

2.2 Defining element types

The model uses element shell163, Fig.5. Shell163 is a 4-node element with both bending and membrane capabilities. The element has 12 degrees of freedom at each node: translations, accelerations and velocities in the nodal x, y and z directions and rotations about the nodal x, y and z-axes. This element is used for explicit dynamic analysis only [8].

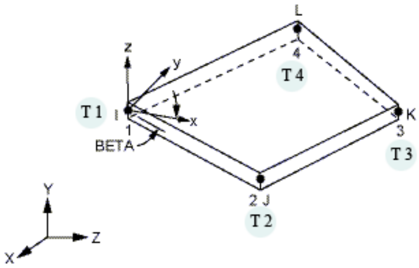

Fig.5 Shell163 geometry[8].

2.3 Real Constants

Real constants are additional properties required to fully define the behavior of the element type that has just been selected (shell element). The values of real constants for the "Belytschko-Tsay" element are: Shear correction factor (SF) = 1, blank thickness = 0.5 mm, number of integration points = 5.

2.4 Choosing the material models.

The proposed study uses material model of type "plastic kinematic". The mathematical equation of this material is [6]:

$$\sigma_y = \sigma_0 + E_p . \varepsilon_p^{eff} \qquad (1)$$

Where σ_y is the yield stress, σ_0 is the initial yield stress, E_p is the plastic hardening modulus which is given by $E_p = \frac{E_{tan} . E}{E - E_{tan}}$ where E_{tan} is the tangent modulus, ε_p^{eff} is the effective plastic strain. Table (1) shows the material properties of the blank.

Table (1): Material properties for the blank

Steel	AISI 1008– low carbon steel
Carbon percentage	C = 0.06 %
Density	7800 kg/m^3
Young's modulus	216 GPa
Poisson's ratio	0.3
Yield stress	216 MPa
Tangent modulus	500 MPa
effective plastic strain	0.3

The material used for the tools (punch, blank-holder, die) is of the type "rigid body". Using rigid bodies to define stiff parts in a finite element model can greatly reduce the computation time required to perform an explicit analysis.

2.5 Model meshing

Meshing a part is the process of generating the chosen elements for that part, for 3D parts, surface or volume mesh may be used. A surface mesh will generate the elements on the surfaces of that part, while the volume mesh generates elements inside the volume of the part besides the outer surface [6,8]. It is important to achieve a mesh with uniform element sizes (i.e., avoid areas with relatively small elements). A large difference in element sizes can cause a small minimum time step size ( ) and, therefore, a long run time. Figure.6 shows the meshed half finite element model of punch , blank, blank holder, and die.

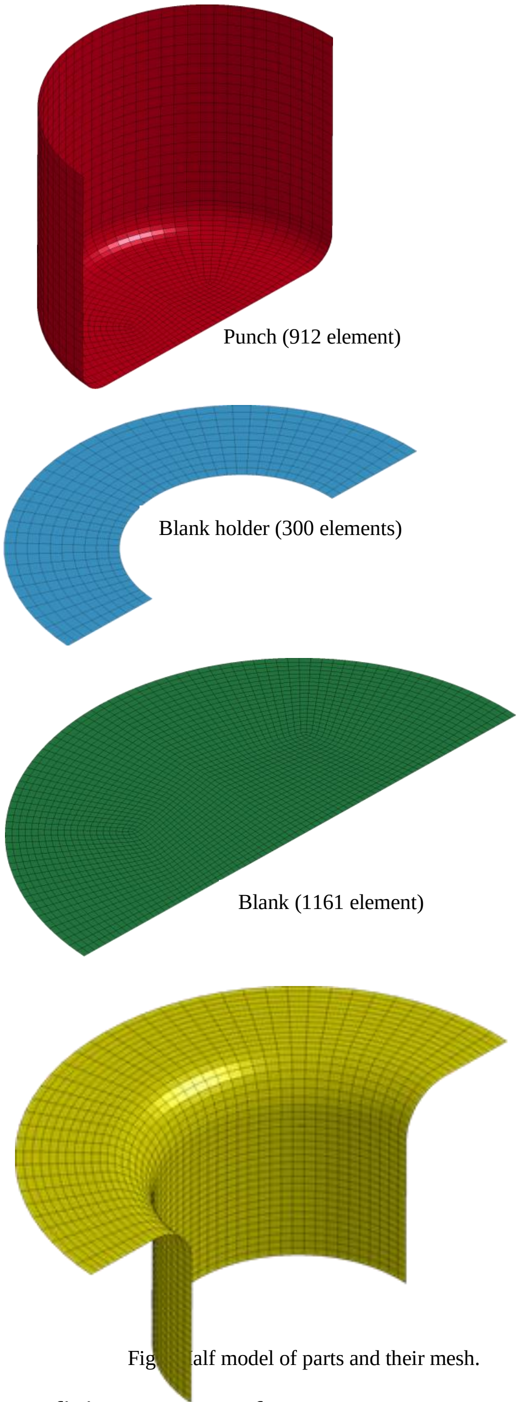

Fig alf model of parts and their mesh.

2.6 Defining contact surfaces

The current model uses surface-to-surface (STS) contact type in which the contact is established when a surface of one body penetrates the surface of another body [9]:

2.7 Applying loads and boundary conditions

After building the model, loads must be applied to the structure in preparation for solution. In order to properly model the structures behavior, it is necessary to apply loads with respect to a specified time interval. Unlike most implicit analysis, all loads in an explicit analysis must be time-dependent in nature. Below are the parts and boundary conditions applied to them.

i. The blank part

Since a quarter FE model is used to represent the tools, both of the blank sides must have a symmetry boundary condition. The symmetry plane should not move in the direction of its normal and should not rotate around the axes that make that plane, but it is allowed to rotate around its normal axis. Figures.7 and 8 show the symmetry boundary condition and finite element model for the blank.

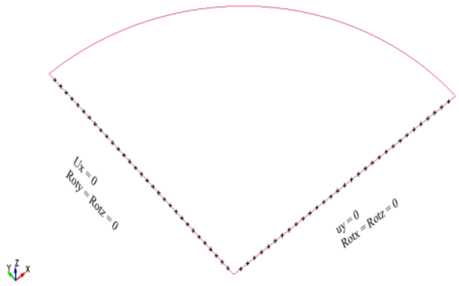

Fig.7 symmetry boundry conditions on blank edges

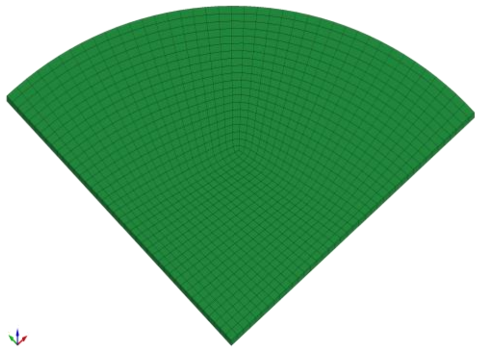

Fig.8 FE model of the blank.

ii. The blank holder part

To prevent wrinkling, the pressure needed for various materials can be calculated as follows **[10]**:

$$p = \frac{R_m}{400}\left[(\beta - 1)^2 + \frac{d}{200.s}\right] \qquad (2)$$

where p is the blank holder pressure (N/mm^2), R_m is the tensile strength (N/mm^2), β is the actual drawing ratio $\beta = (D/d)$, *D* is the blank diameter(mm), d is the punch diameter(mm), s is the sheet thickness(mm).

iii. The die part

The boundary conditions applied are: zero displacement in all directions and zero rotations in all direction.

iv. The punch part

The boundary conditions for the punch are: all nodes rotations are zero, $Rot_x = Rot_y = Rot_z = 0$, Only vertical displacement is allowed, $U_x = U_y = 0$, a prescribed velocity load curve, Figure.9, which contains downward velocity values of the punch versus time is applied to the rigid part that represents the punch[11].

3. Results and Discussion

The developed finite element model is validated prior to implementation by utilizing the cup thickness as an indicator of failure.

3.1 Validation of the FEM model

Two 3D explicit finite element models were used to validate the finite element method results. (a) The first model has a punch profile radius of (P = 4 mm), a die profile radius of (D = 8 mm), (b) The second model has (P = 6 mm) and (D=8 mm), for each of the two models three blank diameters are considered (B= 78, 82, 86 mm) respectively.

The validation method depends on comparing the results of the thickness obtained from the numerical method with those obtained experimentally by the researchers.

Figures.10-A to 10-C and 11-A to (11-C) showed a comparison between the numerically obtained thickness distribution values along the cup cross section from the center of the cup bottom to the outer rim edge with the experimentally obtained values for a model with constant die and punch profile radii (p= 4, 6) mm, (D = 8) mm with three blank diameters (B=78, 82, 86) mm.

By observing each of those three Figures, the noticed behavior of these curves are almost similar in shape but there is a difference in the values, generally, this difference increases when going from the center of the cup to the outside edge of the flange. Also four regions along the cup section are identified:

i. There is a region which has a small thinning; it is located under the bottom of the punch face. This area does not undergo any deformation so remaining in its original flat shape.

ii. The second region which is in contact with punch corner has a larger thinning due to the stretching of the metal in that area.

iii. The third region is the vertical cup wall area which increases in thickness towards outer cup edge.

iv. The flange rim (the area under the blank-holder) has maximum thickness.

The maximum thickness error values for figures.10-A to 10-C are as follows: Figure A has an error of (8%), Figure B has an error of (9.9%), Figure C has an error of (13%). When the error value converge the maximum error limit, this indicates that the blank diameter should be reduced. It is noticed that the error increases with the blank diameter.

3.2 Effect of friction

The results show that any change in (μ) will alter the predictions. The higher friction restrains the movement of the blank, producing more localized strain profiles and more punch force. Figure.12 shows the effect of friction on cup wall thinning.

3.3 Effect of Blank Holder Force (BHF)

For BHF= (1.25) kN the blank gradually loses the contact with the flat part of the die and helps in increasing the distance between the blank holder and the die. This effect is likely to be the reason for wrinkling occurrence at the cup wall. For BHF= 2.5, 5 KN also appears a distance between the flat face of die and the blank holder. For BHF= 10 KN the flat face of the die is remained contact with the blank holder at all steps. Since force of 10 KN gives a minimum wrinkles it was used in FE simulation. Figure.13 represents the effect of BHF on thickness distribution of the cup wall, it is seen from the figure that the thinning at the region of flat bottom face has a value increases with increased BHF.

3.4 Effect of Die profile radius

Thickening increases with increasing die corner radius and maximum thinning occurred at the smallest die corner radius (D = 2). The effect of the larger die profile radius is to reduce the amount of stretching which occurs over the punch profile radius of the cup. Figure.14 shows die profile radius effect on thickness.

3.5 Effect of punch profile radius

The thinning at the flat bottom punch region has a value that increases with decreasing punch profile radius. Figure.15 shows effect of punch profile radius on thickness distribution.

4. Conclusions

From the present work results, the following conclusions can be drawn:

1. A 3D finite element cylindrical deep drawn cup can be simulated using ANSYS, LS-DYNA. The adopted model showed a reasonable agreement with experimental work of some researchers with a maximum error of 15%.
2. The failure of the deep drawn cup can be predicted by relying on the numerically obtained values of thickness since they match the referenced experimental values for blank diameters of 78 and 82 mm.
3. The thickness distribution variation showed that there is always a difference in numerical predictions, since no model can entirely capture all the complex parameters of the real deep drawing process.
4. The model was found sensitive to friction, indicating that a change in friction coefficient (μ) will alter the predictions. It was noticed that a higher value of μ restrain material movement, producing more thinning and more punch force.
5. An increase in blank holder force will decrease the thickness of both the bottom face of the cup and the flange rim.
6. The thickness increased with increasing die corner radius and maximum thinning occurred at the smallest die corner radius.

7. The thickness at the flat bottom punch region decreases with decreasing punch profile radius.

5. References

[1] Amit G., "Blank Optimization in Sheet Metal Forming Using Finite Element Simulation", M.Sc. Thesis, Texas University, Mech. Eng. Department, US, 2004

[2] Z. Deng, W. Bleci, M. R. Lovell "A General Approach for Predicting the Drawing Fracture Load and Limit Drawing Ratio an Axisymmetric Drawing Process", Metallurgical and materials transactions, Vol. 30A, October, 1999, P. 2612.

[3] Hong Yao, Brad L. Kinsey, Jian Cao "Rapid design of corner restraining force in deep drawn rectangular parts". Transaction of the ASME vol.123, 2001, p.248-257.

[4] Hun .H and Kim .S, "Process Design For Multi-Stage Elliptic Cup Drawing With The Large Aspect Ratio", Journal of Material Processing Technology, Vol. 113, Issue 2, 2001, p.779-785.

[5] Khalil E. Abass, "Prediction Of Deep Drawing Ratio By Using FEM", M.Sc. Thesis, University of Technology, Bagdad, Iraq, Production Eng. Department , 2006.

[6] ANSYS Software Help-Ansys LS-Dyna user guide, Livemore Tech. Inc., 2008.

[7] Ivana S., "Handbook of Die Design", 2nd Edition, McGraw Hill Pub., 2006.

[8] ANSYS Modeling and Meshing Guide,ANSYS, Inc. Southpointe, 275 Technology Drive, Canonsburg, PA 15317

[9] Amit G., "Blank Optimization in Sheet Metal Forming Using Finite Element Simulation", M.Sc. Thesis, Texas University, Mech. Eng. Department, US, 2004.

[10] Bradley N. Maker, Xinhai Zhu, "Input Parameters for Metal Forming Simulation Using LS-Dyna", Report, Livermore Software Tech. Corporation, 2000.

[11] John G. lenard, "Metal Forming Science and Practice", 1st Edition, Elsevier, 2002.

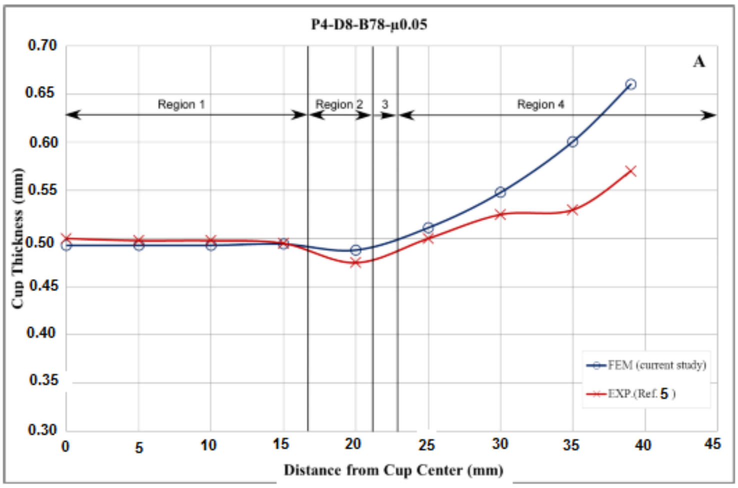

Fig.10-A: Thickness distribution for blank dia.78 mm with (P4-D8) mm.

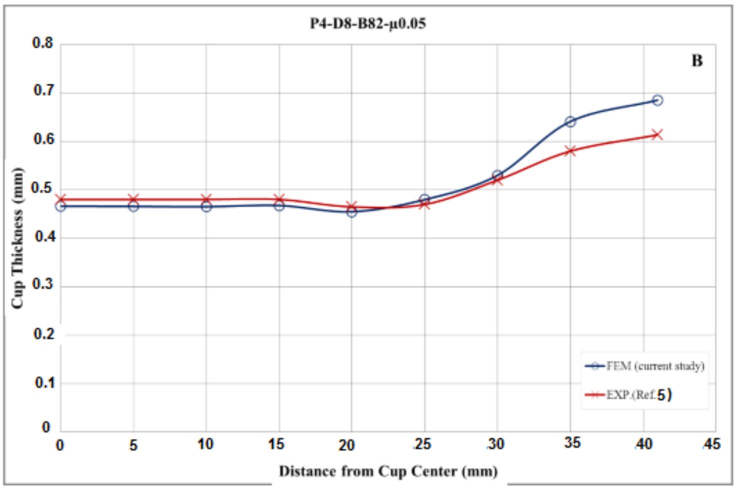

Fig.10-B: Thickness distribution for blank dia.82 mm with (P4-D8) mm.

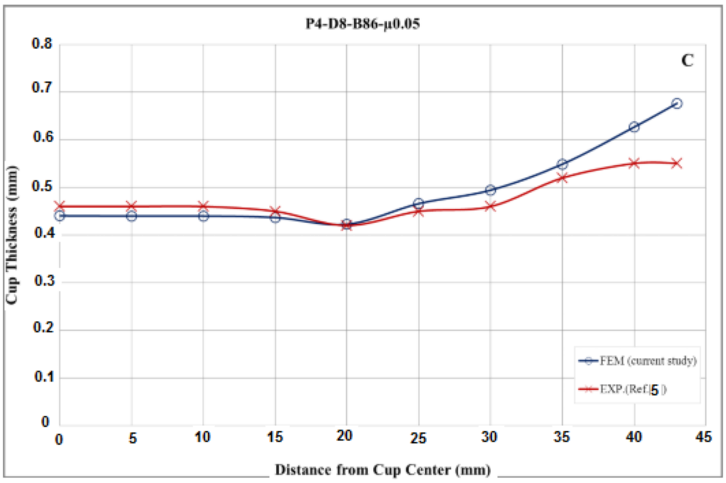

Fig.10-C: Thickness distribution for blank dia.86 mm with (P4-D8) mm.

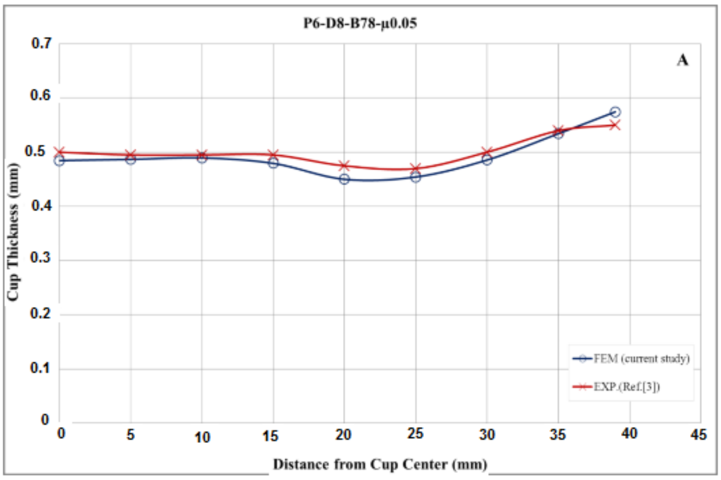

Fig.11-A: Thickness distribution for blank dia.78 mm with (P6-D8) mm.

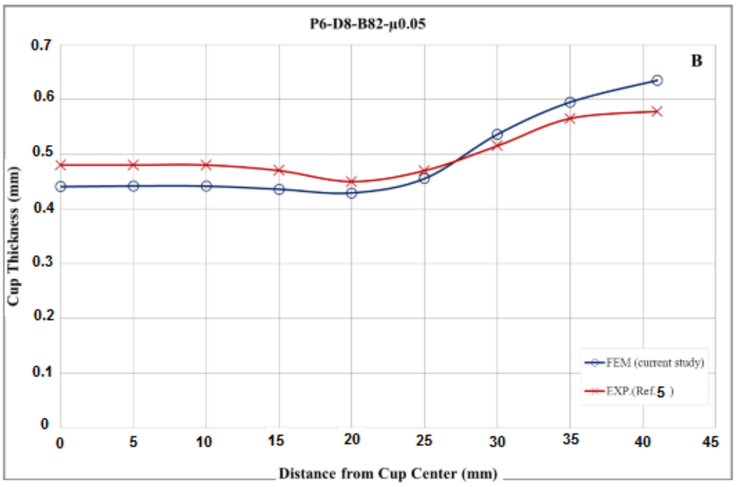

Fig.11-B: Thickness distribution for blank dia.82 mm with (P6-D8) mm.

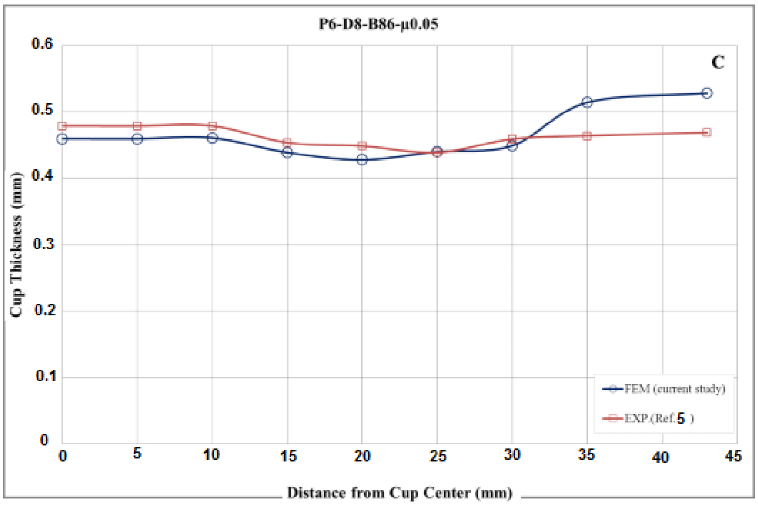

Fig.11-C: Thickness distribution for blank dia.86 mm with (P6-D8) mm.

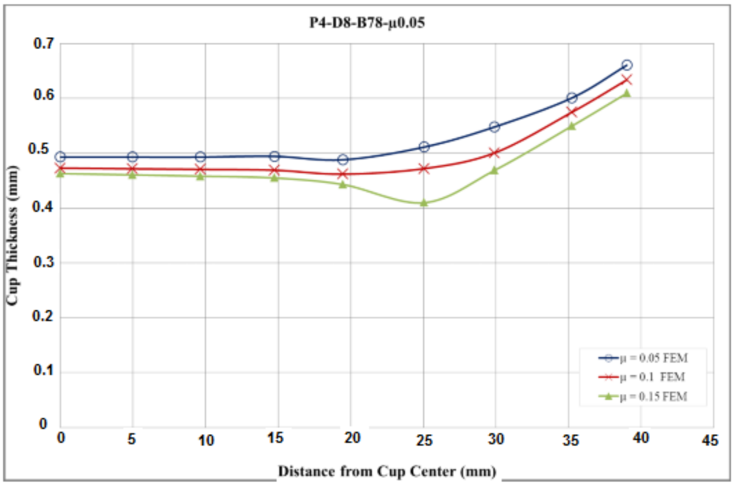

Fig.12 Effect of friction on cup thickness.

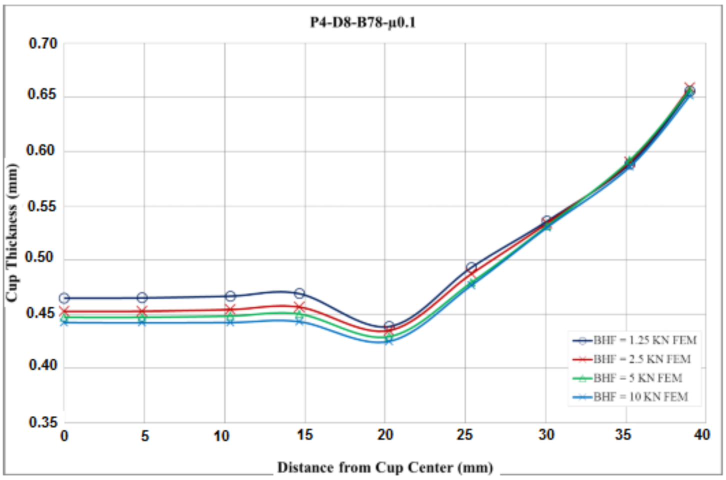

Fig.13 Effect of BHF on thickness distribution

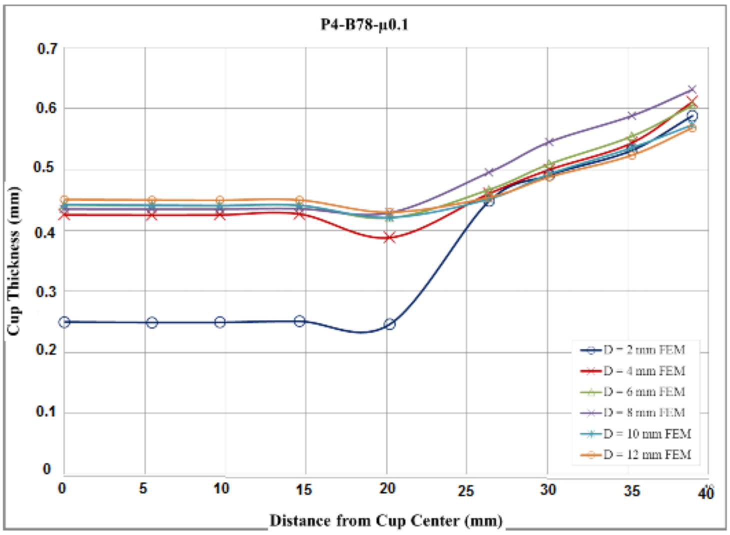

Fig.14 Effect of die profile radius on thickness distribution.

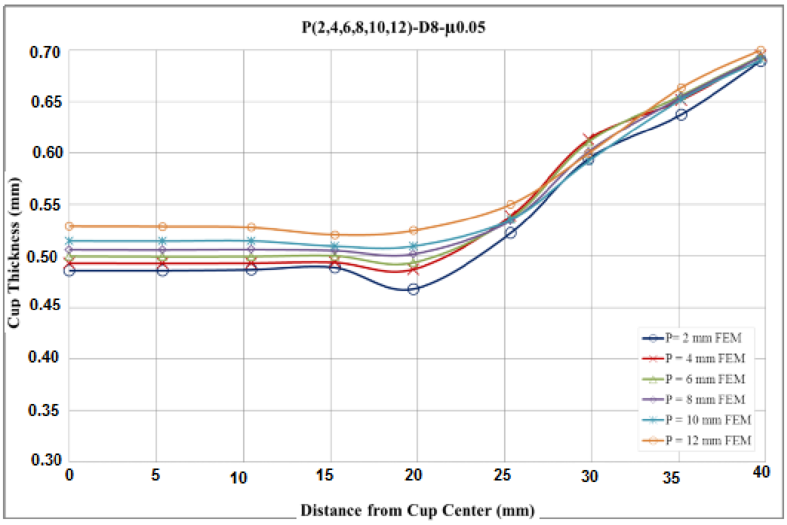

Fig.15 Effect of punch profile radius on thickness distribution.