Eng. & Technology, Vol.25, No.3, 2007

Investigation of Contact Interface Between the Punch and Blank in Deep Drawing Process

Dr. Waleed Khalid Jawad*

Received on: 23/6/2005
Accepted on: 23/11/2005

Abstract

In this work a commercially finite element program code (ANSYS 5.4), is used to perform the numerical simulation of the deep drawing operation.

A simplified axisymmetric model of cylindrical cup of (44mm) outer diameter, (28mm) height and (0.5mm) sheet thickness of annealed mild steel of 0.15% carbon content, has been developed, and the numerical results are compared with the experimental work. Six types of punches of (43mm) diameter with punch profile (nose) radius of {P=3,6,9,15,18, & 21.5 mm}have been constructed and used, and the value of die profile (nose) radius is kept constant to (d=6mm).

This work aims to study the effect of punch profile radius on the interfacial contact between the punch and the blank, punch load, thickness variation over the produced cup wall, localized strains and stresses distribution across the inner and outer wall of the drawn part, the height and amount of spring back of the drawn part.

The results show that ;

The length of contact distance between the blank and the punch increases as the punch nose increases and its value approximately is equal to punch nose radius. Increasing the punch profile radius leads to increasing the cup height about (20 % for FESimulation & 18 % for experimental work), and increasing the value of springback to about (1.75 % for FE Simulation &1.25 % for experimental work)) for punch nose radius ranging from (3 to 21.5mm). The greatest thinning is seen to occur with spherical punch due to great stretching of the blank over the punch head. The punch load decrease slightly with increasing punch nose radius .The more generous punch radius (spherical nose), the more gradual rise of the punch load and larger the punch travel. The stress and strain distributions for all geometries chosen are similar in shape, and have the same trend and approximately the same values for both inner and outer wall of the drawn part.

الخلاصة

تم في هذا البحث استخدام برنامج (ANSYS 5.4) الخاص بتقنية العناصر المحددة في تمثيل عملية السحب العميق. حيث تم تمثيل العملية باستخدام نموذج متماثل المحاور على شكل قدح أسطواني بقطر خارجي (44mm) وارتفاع (28 mm) وسمك (0.5mm) ولمعدن الصلب الطري بنسبة كربون (0.15%) ، مع إجراء مقارنة بين النتائج العملية والنتائج النظرية. وقد تم تصنيع واستخدام ستة انواع من المخارم (Punch) ذات نصف قطر تقوس للحافة مقداره (3,6,9,15,18,21.5 mm) ,في حين ان نصف قطر تقوس حافة القالب الانثى (Die) بقى ثابت بمقدار(6mm)

*DEPT. OF PRODUCTION & METALLURGY ENG, UOT., BAGHDAD, IRAQ.

يهدف البحث الى دراسة تأثير تغير نصف قطر تقوس حافة المخرم على منطقة التماس بين المخرم والصفيحة , قوة ضغط المخرم , تغير السمك على جدار القدح المنتج , توزيع الآجهادات والانفعالات على الجدار الداخلي والخارجي للقدح المنتج , مقدار ارتفاع القدح والارتداد الخلفي (springback) للقدح المنتج.

وقد اثبتت النتائج مايلي:

يزداد طول منطقة التماس بين المخرم الذكري والصفيحة مع زيادة نصف قطر تقوس حافة المخرم , وان قيمتها تساوي قيمة نصف قطر التقوس تقريبا. تؤدي زيادة نصف قطر تقوس حافة المخرم الى زيادة ارتفاع القدح المنتج في حدود (20 %) نظريا , وفي حدود (18 %) عمليا. كما يؤدي الى زيادة قيمة الارتداد الخلفي بمقدار (1.75 %) نظريا , و (1.25 %) عمليا. أن أقصى قيمة لترقق المعدن تحدث عند استخدام مخرم كروي الرأس بسبب المط الزائد للمعدن على وجه المخرم . كما تؤدي زيادة نصف قطر تقوس حافة المخرم الى تقليل قوة السحب بدرجة بسيطة , حيث تكون الزيادة الحاصلة في قوة السحب مع استمرار الشوط بصورة تدريجية وتستغرق مسافة أطول للشوط. آما طريقة توزيع الأجهادات والانفعالات ولجميع المخارم المستخدمة,فقد كانت متشابهة ومتقاربة من حيث القيمة, سواء على الجدار الخارجي أو الداخلي للقدح المنتج.

Introduction

Deep drawing is the metal working process used for shaping flat sheets into cup-shaped articles such as bathtubs, shell cases, and automobile fenders. This is done by placing a blank of appropriate size over a shaped die and pressing the metal into the die cavity with a punch which simultaneously transferring the specific shape of the punch and die to the blank. Generally a clamping or hold-down pressure is required to press the blank against the die to prevent wrinkling and to control the material flow into the die cavity. This is best done by means of a blank holder or hold-down ring in a double-action press.

Rapid developments in computer hardware make the finite element method (FEM) of complex deformation responses increasingly applicable. The FEM is used worldwide to simulate the deep-drawing process and has become a reliable numerical simulation technology, therefore the current exposition here will focus only on the papers concerning the use of FEM in deep-drawing process.

Tremendous progress has been made since the pioneering work of applying FEM to analyze sheet stamping processes, carried out by Wang & Budiansky, (1978)[1], Key & Bath,(1979)[2], who used the elasto-plastic material model, and Kobayashi & Kim,(1978)[3], who used the rigid-plastic material model.

Luca et al. (1996)[4], have presented a comprehensive study of the forming of a sikkens component using two carbon fiber reinforced thermoplastic materials (CFRTP); namely, APC2-AS4 & PEI-CETEX. Both experimental and simulation techniques have been used to understand the formability of the two materials under different processing conditions. The various forming parameters examined have included the forming velocity and the use of a blank holder clamping system. Experimental and simulation results (using PAM-STAMP software) have demonstrated that a low punch velocity will give improved formability and less likelihood of wrinkling.

Meguid & Refaat (1997)[5], have examined the effect of interfacial friction on punch load, the Von-Mises stress trajectory, the successive deformed geometry, the springback, and the residual stress in an aluminum sheet. The result show that, the magnitude of springback is influenced by the amount of plastic deformation and unloading residual stress, and the maximum residual stress occurs in regions experiencing maximum plastic deformation.

In (1998), Zhou et al. [6], investigated the effect of friction on the development of ears in FCC metals using FE program ABAQUS. The simulation demonstrated that, friction between the blank-holder and the blank and the die and the blank, decreases the ear heights along the rolling and diagonal direction, but increases them along the traverse direction.

The use of drawbead model was performed by Chen & Chiang (1998)[7], to control the metal flow and to determine the optimum blank-shape of a motorcycle oil tank, as well as to prevent fracture and wrinkle in the early design stage. The results show that draw bars in the wrinkled area absorb the redundant metal and lead to obtain a production free from defects.

In (2000), Yao & Cao [8], have 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.

Altan (2000)[9], has verified that FEM could predict the defects that occurred in various forming stages of deep-drawing operation. The results show that wrinkling occurs along the flange during the first deep drawing operation. Fracture was observed along a sharp corner at the second deep drawing operation. This was predicted by a high value of the thinning, above 30%.

Predicting the onset of the wrinkling in aluminum square cup forming, was investigated by Xi Wang & Cao (2000)[10]. It was found that the stress distribution is not uniform and the hoop stress is even not completely compressive in the frustum region (side wall). Those cases demonstrate that the presented analysis provides a simple and effective way to predict the onset of side-wall wrinkling.

The crucial problem in deep-drawing is to determine the optimal tool geometry in order to achieve a desired shape for the product. This problem was investigated by Ghouati et al. (1996,2000) [11,12], they have proposed a numerical procedure for the design of deep-drawing process. It is based on the coupling of an optimization technique and the FEM. It was found that thickness distribution for optimal value is indeed more homogenous than those are for the upper and lower bound on the parameter.

Cao et al. (2001)[13], have developed a new forming process that can reduce forming steps of an axisymmetric thick metal drawing problem. The results show that a measurement of tearing potential is about 5% lower than that in the original one, while the press loads are almost identical. The 10-step drawing is reduced to 6-step drawing.

Huh & Kim (2001)[14], introduced an optimum design procedures to seek the optimum process parameters in sheet metal forming process. Conditions of the blank holding force, the die shape, and the bead force were optimized to obtain a specific quality of products. Results show that the optimum design of process parameters has been well performed to decrease the amount of strain that preventing fracture by tearing.

Jian & Shi (2003)[15], have applied FEM to improve the design of a progressive-die for deep drawing applications. It was found that integration of simulation and past experience could reduce the number of die-tryout tests and associated time and cost.

Ahmed et al. (2004)[16], proposed finite element simulation of axisymmetric sheet forming operations based on the viscous shell formulation. The analysis can predict the strain variation and the deformation history of axisymmetric sheet forming operation.

Numerical Simulation :

A cup of (44mm) outer diameter, and (28mm) height, was chosen for detailed analysis of deep drawing operation. The blank from which it is formed has a diameter of (82mm), (0.5mm) thickness and is comprised of annealed mild steel of 0.15% carbon content, (200MPa) yield stress, (200GPa) Modulus of elasticity, (0.5GPa) Tangent modulus and of (0.3)Poisson’s ratio .This particular cup actually has no flange; it is completely drawn in to the die.

In this work, a commercial FE code (ANSYS 5.4) is used to simulate the deep drawing operation, where a simplified axisymmetric model of cylindrical cup with six types of punches profile (nose) radius of{P=3,6,9,15,18,21.5mm} was created, and the value of die profile (nose) radius is kept constant to (d = 6mm).

The blank material is modeled with (Visco106 element). It is assumed that the punch, die and the blank holder are rigid and are represented by (solid element 42). The contact interface between the die and the deformed material is represented by (contact element 48). A Coulomb friction law was employed to investigate the effect of friction at the tool – material interface (μ=0.1). The blank holding force was ranging from (10- 15 KN) during the full stroke of operation. The clearance between punch and die was set to be (1.1 sheet thickness). Two examples of deep drawing operation using FE Simulation are shown in figure (1). Drawing force (Punch load), length of contact distance, thickness along the wall of the produced cup, height of produced cup, amount of springback for the produced cup, stress and strain distributions over the outer and inner wall of the produced cup was measured. The numerical results were compared with the experimental work.

Experimental Work :

In this work a deep drawing tooling was designed and constructed to carry out the experimental work as shown in figure (2). Six types of punches of (43mm) diameter with punch profile (nose) radius of {P=3,6,9,15,18, &21.5 mm} were used, and the value of die profile (nose) radius is kept constant to (d=6mm). The experiments were carried out using the Instron testing machine which has a capacity of 100KN and cross head speed of

(300mm/min), with punch stroke ranging from (35-40mm). The material properties are the same as that mentioned before in section of the numerical simulation.

In order to foresee the contact interface between the punch and blank during drawing operation, the punches were painted with white color. After completing the drawing operation it was found that the white paint was removed from the punch nose (as result of the blank bending and sliding over the punch nose) and stacked at the inside nose of the drawn part (produced cup). In order to measure the drawn part wall thickness and the length of contact distance; the drawn part was divided in to two parts by using a diamond saw. The length of painted distance, which was removed from the punch head and stacked at the inside nose of the divided drawn part, was measured as shown in figure (2). During the experiment, drawing force (punch load) was recorded along with punch stroke, thickness of the produced cup wall, length of contact distance, height of produced cup, and the amount of springback for the produced cup was measured and compared with the results obtained by the simulation.

Results & Discussion:

A successive stages of the deep drawn blank for varying punch strokes are shown in figure (1).It is shown obviously that relatively high stress concentration occurred at the die corner radius as a result of excessive bending of material. The material lying under the flat punch is more constrained, and therefore deformed less. and localized thinning occurred near to the punch corner. This mean that frictional force is applied to the metal largely by the edge of the punch and not by its flat section. Therefore, it has been assumed that friction force is present wherever there is a curvature on the surface of the drawn part. Therefor for spherical punch (P=21.5mm), material laying at the punch nose deforms more than the other portion of the cup wall and produces more thinning leading to high stress concentration at the punch (nose) radius as a result of excessive bending and stretching of metal over the punch nose.

Figure (2) show a comparison between FE Simulation and experimental work for the length of contact distance between the punch and the blank . It is seen obviously that the contact state is focus at the curved portion of the punch or at the curved portion of drawn part as a result of excessive bending and sliding of metal over the punch nose. The length of contact distance increase as the punch nose increased and its value approximately equal to punch nose radius, this is the reason why some times a hollow punches with definite nose radius are used in deep drawing dies.

In the metal forming operation, the highly nonlinear deformations processes tend to generate a large amount of elastic strain energy in the materiel besides of the same plastic deformation area. The elastic energy, which stored in the metal sheet during the forming, is subsequently released after the forming pressure is remove. This release of energy is the driving force for spring-back of sheet metal forming. Hence, the spring-back deformation for the sheet metal forming are mainly lie on the amount of elastic energy stored in the part while it is being plastically deformed. There are numerous factors that influence elastic energy stored in metal sheet, such as the shape of the die, the properties of the material, the boundary conditions and the interfacial loads.

In this research, the amount of spring-back was calculated as follow:

Spring-back % = ((measured diameter of the drawn part – Design or nominal diameter of the part) /nominal diameter of the part)) x 100

Figure (3) show a comparison between FE Simulation and experimental work for the effect of punch profile radius on the height of produced cup and the amount of springback. It is shown that increasing the punch nose leading to increasing both the cup height and the value of springback due to excessive bending and sliding of the metal over the punch nose radius and due to excessive stretching of metal over the large punch nose radius. It is found that the values of cup height increasing reach to about (20%for FESimulation &18%for experimental work), and the value of springback to (1.75 % for FESimulation & 1.25 % for experimental work) for punch nose radius ranging from (3 to 21.5mm).

The variations of the drawing force (punch load) with the punch stroke under different punch profile radius are shown in figure (4). It is shown that the drawing force for large punch nose radius is lower slightly than that for small nose radius, in both simulation and experimental work. This can be explained by more sever bending effect on small nose radius. Necking was considered to occur as the punch force reached its maximum value and started to drop. Following necking, splitting occurred at the point of the large drop in the punch force .It is seen that the more generous punch radius (spherical nose), the more gradual rise of the punch load and larger the punch travel.

Thickness of produced cup wall was measured and compared with the thickness obtained by FESimulation as shown in figure (5). It is clear from the figure that initial blank thickness (0.5mm) at the region of a flat bottom face of the punch does not change and remains almost constant, this is because the flat face of the punch is in contact with blank, and due to the drawing force, friction comes into play which prevents any deformation of the metal under the punch. Hence there is no thickness change observed at this region. However, the necessary deformation is provided by other portions of the cup, resulting in an increase of strain in the wall and flange regions. The greatest thinning is seen to occur with hemispherical punch (punch profile radius P =21.5 mm) due to great stretching of the blank over the punch head.

Figure (6) show the Effective (equivalent) strain and Effective stress distribution over the cup wall of completely drawn part for all the punches geometry chosen. It is clear from these figure that the stress and strain distributions for all geometries chosen are similar in shape, and have the same trend and approximately the same values, except for the large nose radius (15,18, &21.5mm), Where high stress and strain concentration are present at the punch nose as a result of excessive stretching of metal over the punch nose. It is clear that the nature of effective strain and stress is tension and reaches its maximum value at the cup rim.

A comparison between the values of stress and strain distribution over the inner and outer wall of the cup for various punch profile (nose) radius obtained by simulation are shown in figure (7). It is clear from these figure that the stress and strain distributions for all geometries chosen are similar in shape, and have the same trend and approximately the same values for both inner and outer wall of the cup.thus one can assume that it is quite enough to measure the strain at the outer wall of the cup only.

Conclusions:

The results show that; the frictional force is applied to the metal largely by the edge of the punch and not by its flat section. Therefore, it has been assumed that friction force is present wherever there is a curvature on the surface of the blank. Therefore high stress and strain concentrations, localized thinning are present at the corners of the drawn part as a result of excessive bending and sliding of metal over the punch nose.

The contact state between the punch and the blank is focus at the curved portion of the of the punch or drawn part. The length of contact distance increase as the punch nose increased and its value approximately equal to punch nose radius, this is the reason why some times a hollow punches with definite nose radius are used in deep drawing dies. Since the numerical simulations match experimental results very well, therefor, the contact state in this research can be found numerically without further experiments.

The simulation and experimental results showed that increasing the punch profile radius leading to increasing the cup height about (20 % for FESimulation & 18 % for experimental work), and increasing the value of springback to about (1.75 % for FESimulation & 1.25 % for experimental work)) for punch nose radius ranging from (3 to 21.5mm).

The punch load decrease slightly with increasing punch nose radius .The more generous punch radius (spherical nose), the more gradual rise of the punch load and larger the punch travel.

The greatest thinning is seen to occur with hemispherical punch due to great stretching of the blank over the punch head.

The stress and strain distributions for all geometries chosen are similar in shape, and have the same trend and approximately the same values for both inner and outer wall of the drawn part. thus one can assume that it is quite enough to measure the strain at the outer wall of the cup only.

References:

[1]- Wang N.M. & Budiansky B., “Analysis of sheet metal stamping by a finite element method”, Journal of Appl. Mech., Trans. ASME,vol 4,(1978),p.45-73.

[2]- Key S.R. & Bath.K. K ,“On the application of finite element method to metal forming processes”, Comp. Meth. in Appl. Mech. and Eng.,(1979),p.579-508.

[3]- Kobayashi S. & Kim J.H., “Deformation analysis of axisymmetric sheet metal forming processes by the rigid plastic finite element method”, Mechanics of Sheet Metal forming, Plenum press, newyork, (1978).

[4]- Luca P., Plefebure J. & Pickett A.K., “Numerical and experimental investigation of some press forming parameter of two fiber reinforced

thermo plastic: APC2-AS4 & PEI-CETEX", Fourth International Conference on flow process in composites materials EPCM96,Aberyswyth U.K,(1996),p.1-35.

[5]- Meguid S.A. & Refaat M.H., "Finite element analysis of deep drawing process using variational inequalities, Journal of Material Processing Technology,(1997).

[6]- Zhou Z., Jonas J., Makinde A. & S.R. MacEwen, "Effect of texture on earring in FCC metal: Finite element simulations", International Journal of Plasticity, vol 14,No5,(1998), p.117-138.

[7]- Chen F.K. & Chiang B.H., "Three dimensional finite element analysis for the stamping of a motorcycle tank", Transactions of the ASME, vol120,(1998),p.770-773.

[8]- Yao H. & Cao J., "Assessment of corner failure depth in the deep drawing of 3D panels using simplified 2D numerical models", Transaction of the ASME vol.123,(2001), p.248-257.

[9]- Altan T., "Forming improving progressive die design by simulation", Stamping Journals, (2000), p.98-101.

[10]- Wang Xi Cao & J., "On the prediction of side-wall wrinkling in sheet metal forming processes", Int. J.Mech. Sci., vol 42,(2000), p.2369-2394.

[11]- O.Ghouati & J.C.Geline, "Sensitivity analysis and optimization of shape and process parameters in the metal forming", Engineering System Design and Analysis Conference, vol 3,(1996), p.221-226.

[12]- Ghouati O., Lenoir H. & Gelin J. C.,"Design of deep drawing processes using the finite element method", Applied Mechanics Laboratory , University of Franche-Comte-Besancon-France, (2000).

[13]- Cao J., Cedric Z. & Tang S.C. , "Analysis of an axisymmetric deep-drawing part forming using reduced forming steps", Journal of Materials Processing Technology, (2001), p.1-18.

[14]- Hun H. & Kim S., "Optimum process design in sheet metal forming with finite element analysis" Journal of engineering material and technology, vol.123,(2001),p.477-481.

[15]- Jian N. & Shi X., "Simulation confirms deep drawing die design" Metal forming magazine,(www.metal forming magazine.com), (2003).

[16]- Ahmed M.., Sekhon.S.G. & D. Singh, "Finite element simulation of axisymmetric sheet forming operations using shell element", (2004), (internet).

[17]- Song Y., Yao D, , Wu S., & P Weng., " Spring-back simulation of sheet metal forming for the HT-7U vacuum vessel " institute of plasma physics,chinese academy of sciences (ASIPP), Email:songyt @ mail.ac.cn.

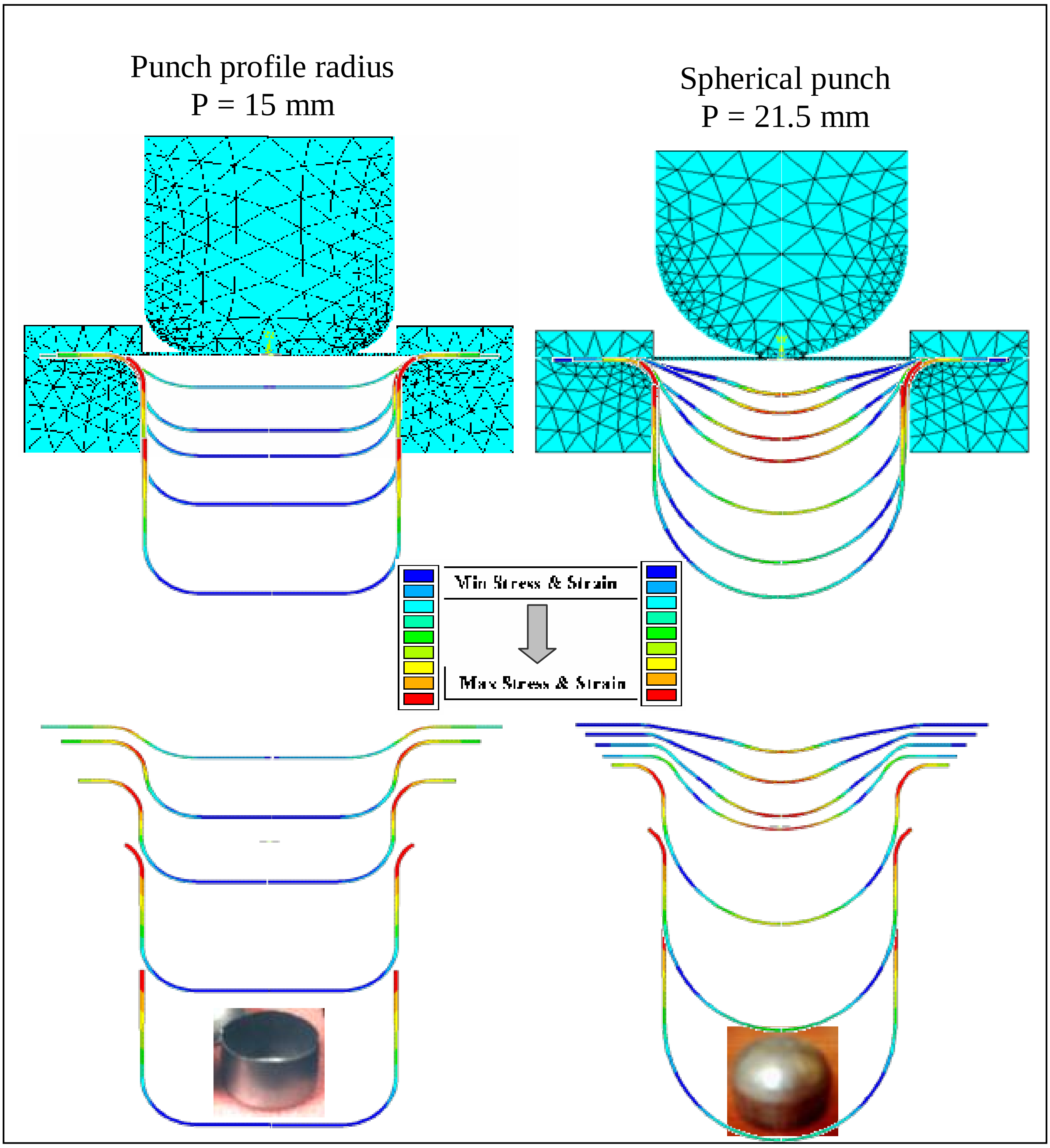

FIG (1): FE Simulation of deep drawing operation for varying punch stroke, showing the effect of punch stroke on stress & strain distribution over the produced cup wall.

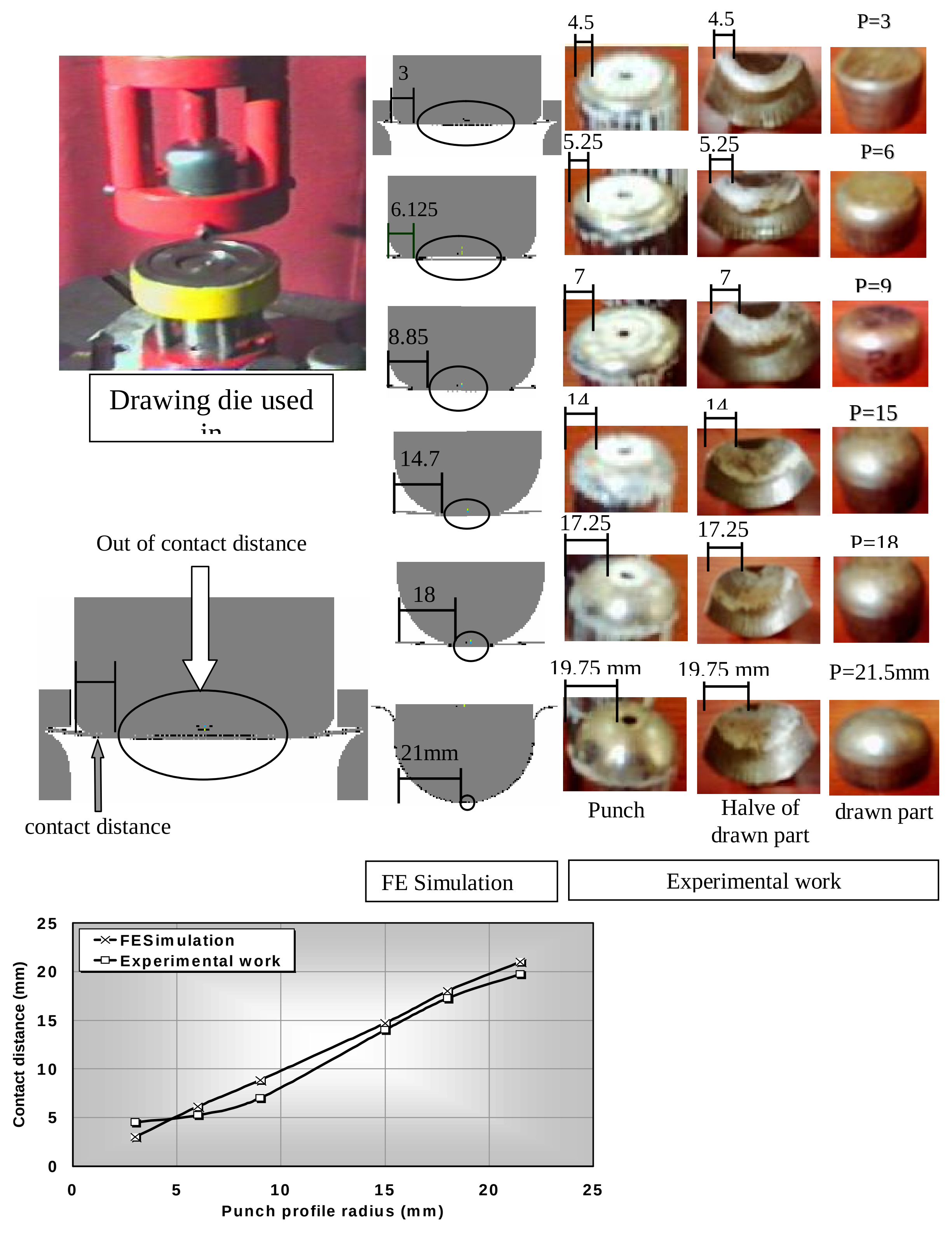

FIG (2): The effect of punch profile radius on length of contact distance between the punch and the blank. (*Comparison between FEM & Experimental work*).

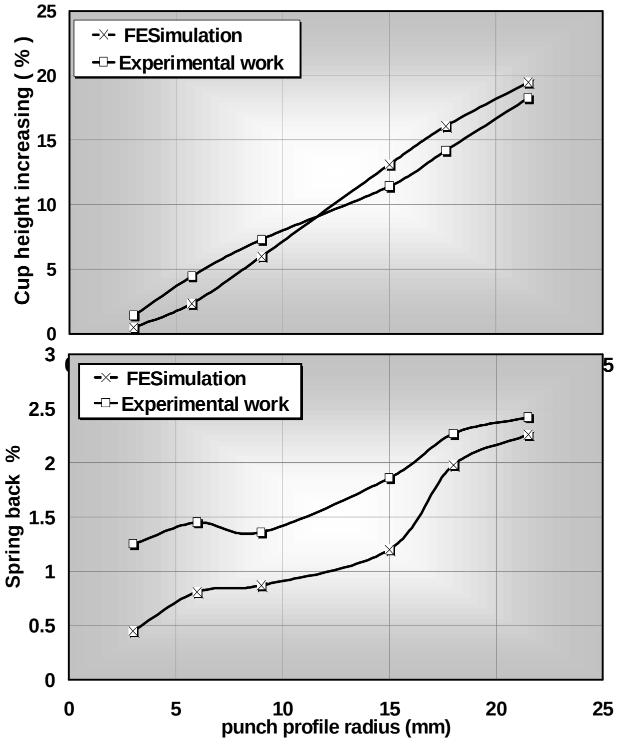

FIG (3): The effect of punch profile radius on the height of produced cup & the amount of spring back.

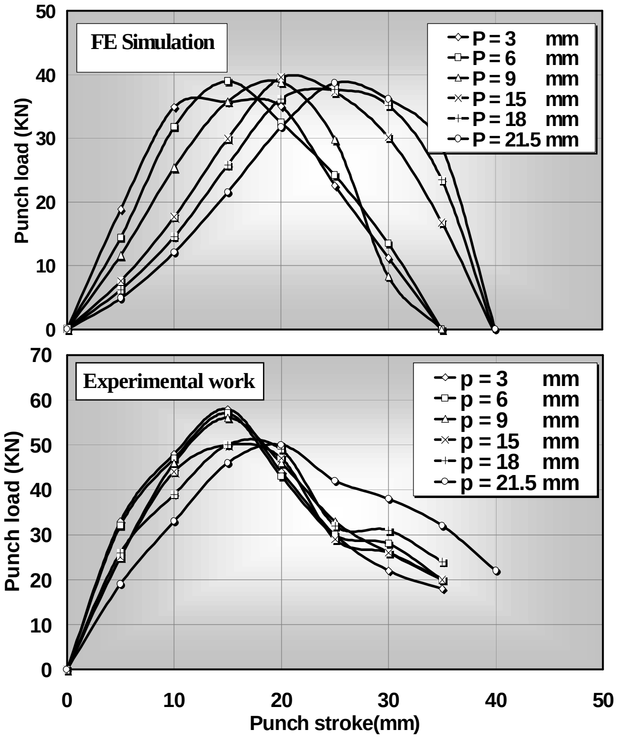

FIG (4): The effect of punch profile radius on punch load.

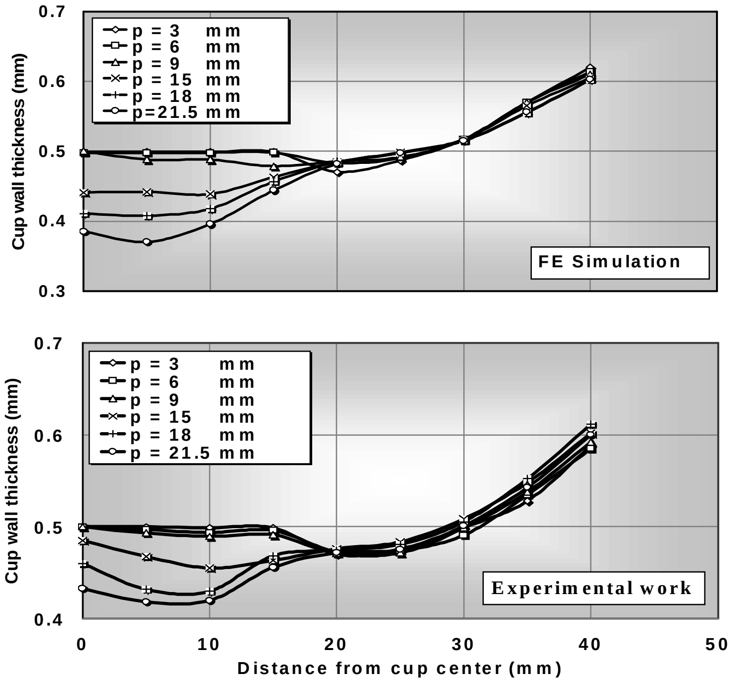

FIG (5): The effect of punch profile radius on cup wall thickness.

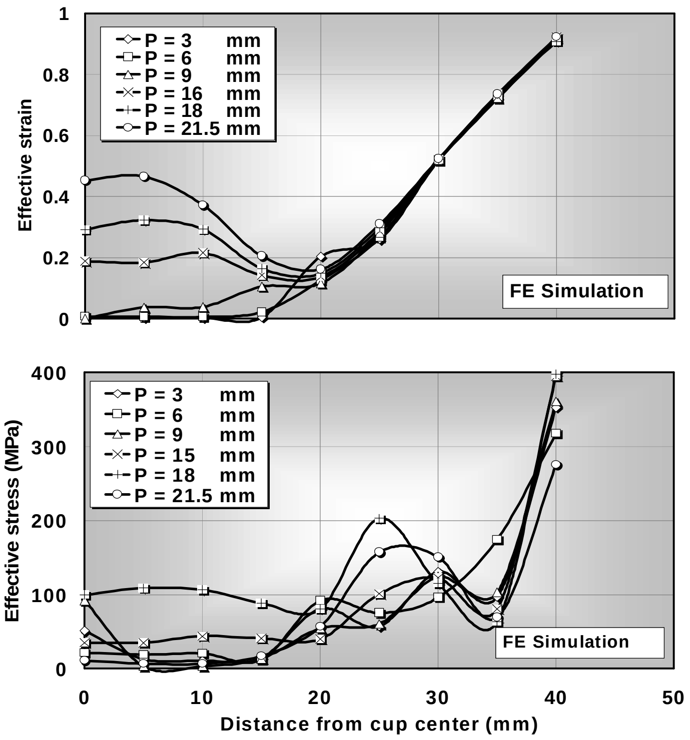

FIG (6): The distribution of stress & strain over the cup wall for different punch profile radius.

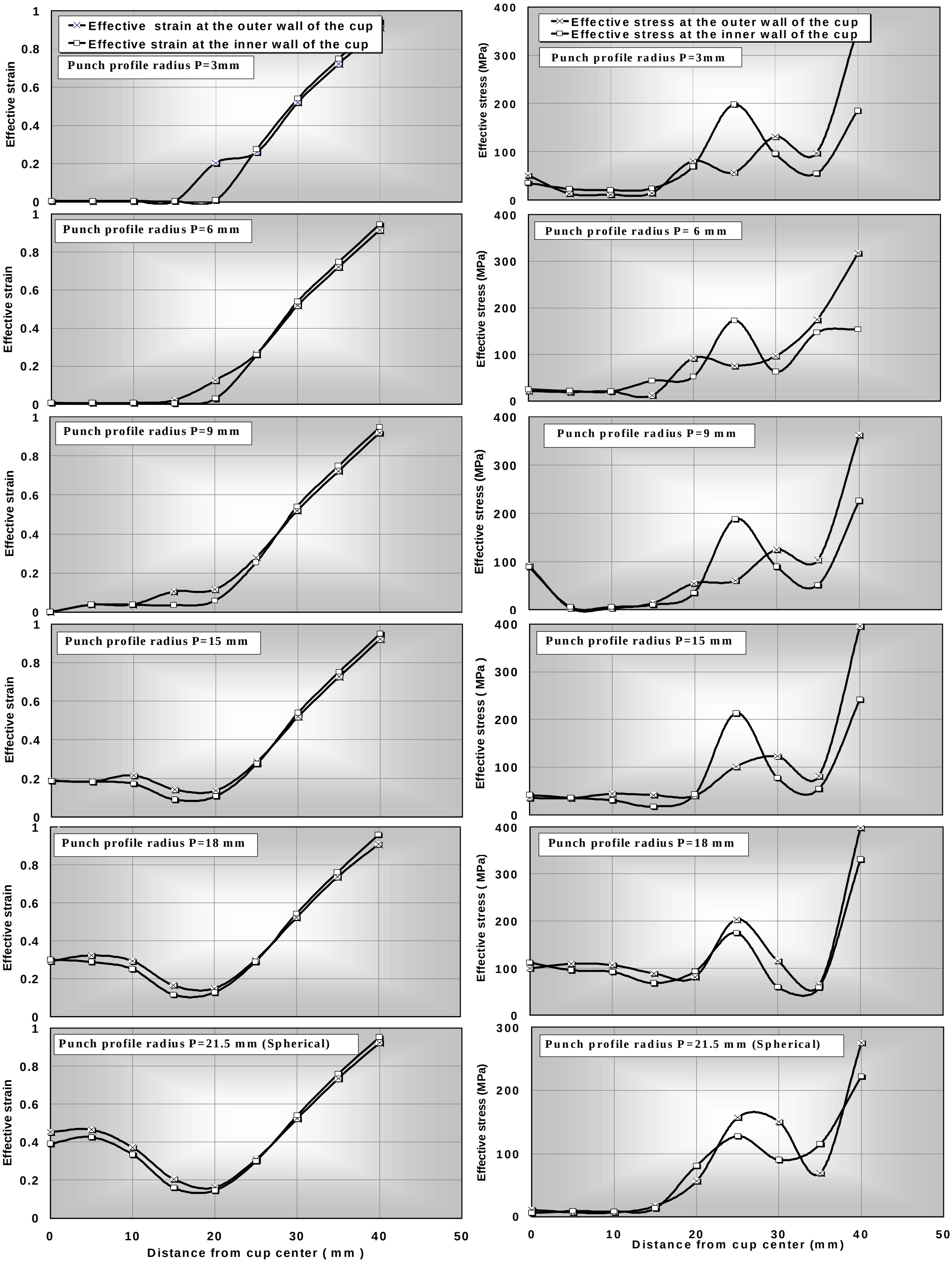

FIG (7): The effect of punch profile radius on stress & strain distribution over the inner & outer wall of the produced cup