Journal for Technology of Plasticity, Vol. 26 (2001), Number 2

FINITE ELEMENT APPROACH IN PLATE BENDING PROCESS

M. Math[1], Z. Keran[2], B. Grizelj[3]
[1,2] Faculty of Mechanical Engineering & Naval Architecture, University of Zagreb, Departement of Technology, Ivana Lučića 5, 10000 Zagreb, Croatia
[3]Departement of Technology, Faculty of Mechanical Engineering, University of Osijek, Ivane Brlić Mažuranić 18, 35000 Slavonski Brod, Croatia

ABSTRACT

In order to determine the inevitable mechanical spring back of a bended plates, designed for assembling the spherical tanks, made of steel StE 500, according to DIN 17102/83, an elastic-plastic incremental finite element calculation has been carried out to analyse axisymmetric strain sheet metal bending process. Sufficiently accurate stress distributions and deformed geometry of plates as parts of spherical tanks have been obtained through the whole bending process. The load-deflection curve has been carried out based on reaction forces and compared with experimentally obtained results. Friction effects and material characteristics such as work hardening are also covered. The accuracy of the simulation results, mechanical spring back and the residual stress distribution after unloading are discussed through the comparison with the experimental results.

1. INTRODUCTION

Sheet metal bending is a standard manufacturing operation in many industrial processes. Despite its broad application, the design of the tools and the selection of the sheet materials are still usually based on trial and error efforts, a very expensive and time consuming procedure. Reliable models for describing the process would be of great value in reducing much of the tool tryout work. The continuum mechanics approach is so complex that a closed - form, exact solution cannot be obtained. Thus, computational methods have to be used. It is useful to simulate the process on the computer before experimental development and die trying - outs are conducted. In such a way, improvements in process conditions and die design are made at the engineering stage before dies are manufactured and expensive production machinery is tied up for costly experimentation. In order to obtain practical results from these computer simulations, and prove that there are relatively accurate, the presented study was conducted and comparison with the experimental data has been made.

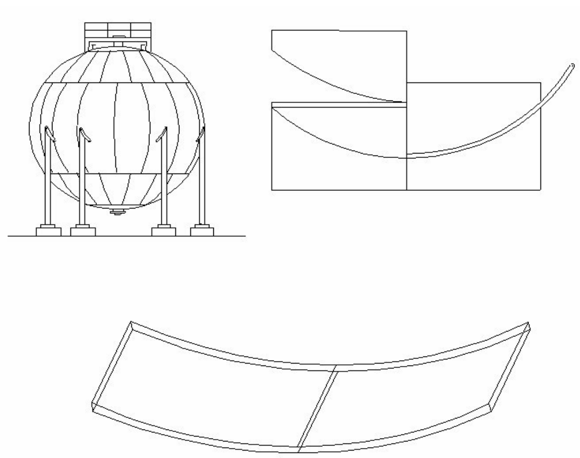

Figure 1. Schematic representation of bending in tool

2. ESTABLISHING THE PROBLEM

The existing increasing necessity for spherical tanks is easy understandable because it exists the possibility to store medium with minimal thickness of tank, small needed volume and minimum cost price. Spherical tanks are becoming far more interesting with increasing of their radius. The shell of spherical tanks consists of steel sheets and upper and down spherical sectors. (see Fig. 1). All parts are produced in cold conditions on hidraulyc press, and then the segments, bended in several indentations are assembled by welding in whole at the actual place. After assembling it is performed hydrostatic probe according to tehnical demands as well as others anticipated by other conditions. The radius of spherical tanks depends on the volume of the stored medium, while thickness depends of the pressure of the medium. Also as a troublesomeing moment it occurrs the tendency that, one tool can be used for several types of tank radii, where average mass of the tool of cast iron is about 30 000 kg (see Fig. 2). Very easy one can understand the wish, that with one tool it is possible to produce as much different plates as possible.

In metal forming processes mechanical springback of bended plates is of great importance. Particularly, it has essential consequences in bending double or spacious curved plate in the case of cold forming. Double or spacious curved plates served for manufacturing spherical reservoirs, water tanks, protective mantles for atomic powerplants, basis of the cylindrical tanks, parts of ship's hulls and similar. In designing the punch and the die it is always necessary to predict the mechanical springback of the plate. Owing to that reason, the radius of the die and the punch is different from the radius of deformed plate of spherical tank or another previously mentioned products. In attainable literature much of the problems are concerned with single curved plate and such cases are widely theoreticaly and practicaly fundamented, but very few associated with in fact spacious bended plates [3]. It has to be emphasized that the actual axial-symmetrical bending analysis is performed with the data used so far in the manufacturing i.e. to deformation of 2%

where elastic part of deformation is relatively great, creating much more problems than in the cases where plastic part is absolutely and relatively greater. The reason for that is very obvious: it is much more difficult to establish phisical phenomena describing a forming operation with quantative relationships in the case of in fact threedimensional stress and strain conditons.

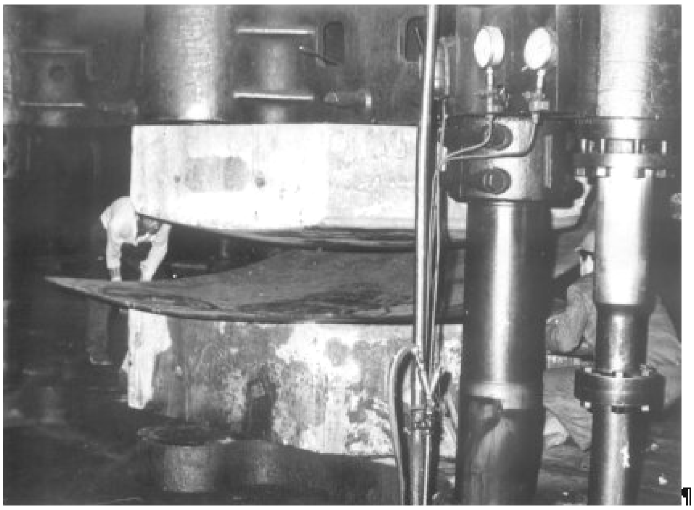

Figure 2. The bending of segment in tool

The metal flow, the friction at tool-material interface, and the relationships between microstructure/properties and process conditions are much more difficult to predict and analyse. For a given operation both preforming or finish forming, such design consists of (a) establishing the kinematic relationships (shape, velocities, strain rates, strains) between the deformed and undeformed part, i.e., to predict metal flow; (b) establishing the limits of formability or productability i.e., determines whether it is possible to form the part without surface or internal failure; and (c) predicting the forces and stresses necessary to execute the forming operation, so that tooling and equipement can be designed or selected. All that is much more difficult when spacious stress and strain conditions take place. For that reason, the actual situation connected with unavoidable mechanical springback is, that one tool is machining several times in intention to establish corect radius causing much time consuming and price increase.

3. EXPERIMENTATION AND SIMULATION

So far the metal segment are deformed as mentioned through several indentations,while mechanical spring back changes the dimension of radius realised by the tool. This presents additional difficulties in assembling, because, in that case it has to be undertaken the slow and expensive dimensional mending. Owing to that reason, the predicting of the amount of mechanical spring back and the knowledge of the radius after releasing the punch is of utmost importance. On that way one can still during the bending process use the tool dimensions, force and punch motion, thus avoided the effects of mechanical spring back and achieve correct radius of spherical segment. Moreover, it exists totally more interesting different tanks which have various diameters, from 800 mm till 21200 mm.

In experiments there are researched several known factors that influence the amount of mechanical spring back:

- the force as it is known, untill some definite value has influence in such a way that greater the force is, the smaller mechanical spring back is induced. But, after some limiting value after overloading there is no sense to increase the bending force, because the amount of mechanical spring back remains unchanged.
- the time of indentation has as it is shown in some works in fact no influence on the amount of mechanical spring back, and that was in acual experiments proved too.
- the influence of material has been investigated, where it is also expecting that material with greater strength and ductility has greater mechanical spring back, but the limitations of the sort of material depends of the medium that has to be stored and the maximum inner pressure demanded by customers.
- also, the thickness of the plate as well as radius of the spherical tanks are limiting possible variations and in fact depend on the designing reasons and takeing into account the strength of design.

Depending on the construction of the die and the punch, the bending of the plate can be treated as a free bending and the bending in the die. There are two different stages of bending. In the first stage, the free bending exists so far unless the sides of sheet are parallel to the sides of the die and touch it (second stage). Until that moment the distance between the supports is equal to the width of the die, and the bending radius is larger then the radius of the punch. In the moment of realisation of the contact between the sides of the sheet and the sides of the die, the scheme of the process is suddenly changed and the distance between the supports is rapidly diminishing, because the places of the contact are removed in the direction of symmetry.

The load - deflection characteristic depends and corresponds to the actual bending stages. The free bending process consists of the elastic part of bending, then the period where the bending force is approximately constantly increasing, and the part when the force increases rapidly, because the calibration is taking place and the bending force and load reach the maximum values.

The numerical analysis was performed using MARC II software. Non-linear analysis always requires incremental solution and sometimes iteration or recycles within each load/time increment to ensure that equilibrium is satisfied at the end of each step. In the presented bending problem, the full Newton-Raphson iterative procedure is chosen to solve the iteration process and non-linear equations of motion. The method has quadratic convergence properties and the stiffness matrix is reassembled at each iteration. This means that in subsequent iteration the relative error decreases quadratically. If material nonlinearities are presented, some approximations slow down the convergence, but these computational problems are less significant when the iterative solvers are used. Since material elements rotate during bending, the finite deformation theory must be utilised for the calculation so that the large displacement, finite strain plasticity and updated Lagrange procedure was adopted. In the updated Lagrangian approach, the element stiffness is assembled in the current configuration of the element, the stress and strain output is given with respect to the coordinate system in the updated configuration of the element while Hill's reference - state - based variation principle can be expressed by the form:

$$\int_v \left\{ \left[\tau_{ij}^{'} - 2\sigma_{ik} D_{kj}\right] \delta\, D_{ij} + \sigma_{jk} L_{ik}\ \delta\ L_{ij} \right\} dV = \int_s t_i^{'}\ \delta\ v_i\ S$$

so that an incremental type elastic - plastic constitutive relation is incorporated into it.

Due to the nature of the physical problem, a non-positive definite solution control is forced. As the large displacements are requested, an additional contribution is made to the stiffness matrix labelled the initial stress stiffness matrix and at the same time, the stiffness is formed using four point Gaussian integration. As default, the analysis program uses the full stress tensor at the last iteration, which in general results in the fastest convergence.

In this work in performing the analysis in the first case the axisymmetric four node isoparametric quadrilateral elements are applied. With this type, it is possible to choose an integration scheme option, which makes the dilatational strain constant through the element. Constant dilatation options are recommended for use in a large strain plasticity analysis, where the method eliminates potential element locking and because conventional elements result in computational results too stiff for nearly incompressible behaviour. For the chosen type of element the interpolation functions have been modified in such way, that shear strain variation can be better represented. In addition, a follow force option has been activated, in intention that the distributed loads calculated are based upon the current deformed configuration.

The work piece material is steel St500 produced according to DIN 17102/83. The work hardening rule is presented as true stress versus the true plastic strain rate. The stress-strain curve in plastic field was established in own experiments using specimens B6x50 according to DIN 50125, while the experimental data was obtained on machine that has computational support to conduct the experiments and its accuracy is 1 as demanded with DIN 51221. By examinations of the specimens the obtained results are:

1. True stress or characteristic of material flow
   $k_f = 588 + 700\ \varphi_v^{0.4}$
2. Modulus of elasticity
   E= 206112 - 206795 [N/mm^2]
3. Poisson's ratio
   ν = 0.31-0.33

The work hardening curve must in the actual case be entered as the true stress versus the logarithmic strain in a uniaxial tension test, where it is supposed that material can be assumed as an isotropic hardening one. The isotropic hardening rule assumes that the centre of the yield surface remains stationary in the stress space, but the size i.e. radius of the yield surface expands, due to work hardening.

Due to the symmetry of the process, just one half of the plate for this first case can be taken into consideration. In order to simulate the previously described bending case the involving boundary conditions assures the full symmetry and are applied in the line of symmetry preventing any move from that line the nodes on the symmetry line are constrained to move in x and y - direction. The curves that define the shape of the die and punch are interpreted as rigid bodies. There are in contact with the plate, which is actually deformable body. Contact between a deformable body and a rigid surface means that the nodes do not penetrate the rigid surface.

When the friction is taken into account the Coulomb friction coefficient is 0.3, and only this case has been experimentally investigated. Furthermore, friction at the interface between bodies can be numerically modelled by one of two methods; by means of any application of distributed shear forces or by direct application of nodal forces. In this work Coulomb friction or adhesive friction is chosen, where friction forces are defined by the following expression:

$$f_t \leq \mu\, f_n\, t$$

Where f_t is tangential force being applied, f_n normal pressure, μ and Coulomb friction coefficient and t tangent unit vector in the direction of the relative velocity

$$t = \frac{V_r}{|V_r|} V_r$$

For the computation of mechanical spring back, the release option is used to enable the releasing of the deformable body's nodes, which have been in contact with the rigid bodies i.e. removing the rigid surfaces, and evaluating mechanical spring back. Either the motion change or contact table option must be used to ensure that the nodes do not recontact the surface they were released from. Two possible ways were examined. In the first one the punch motion option was forced for overload case with approximately 4% of deformation in first examined case, and in the second one, the pressure table option was forced simulating overload with approximately 60% of nominal force as was done in real conditions. The same procedure but using 3D membrane shell elements in intention to have three-dimensional picture of deformation and metal flow as well as possible wrinkling of the edges has been performed. This is four nodes, thick shell element with global displacements and the rotations. Bilinear interpolation is used for the coordinates, displacements and rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at the middle of the edges and interpolated to the integration points. In this way, a very efficient and simple element is obtained which exhibits correct behaviour in the limiting case of thin shells. The element can be used in curved shell analysis and due to its simple formulation when compared to the standard order shell elements, it is less expensive and, therefore, very attractive in non-linear analysis. This element is not very sensitive to distortion, particularly if the corner nodes lie in the same plane. The element thickness is specified in the geometry option. As a representative case, the plate of 13 mm thickness and 460 mm in diameter is chosen. This plate is used for assembling the one of possible spherical tank mentioned in introduction chapter.

4. RESULTS

The Fig. 3 shows the deformed geometry at the end of bending in the case with overloading, while the Fig. 4 shows that in the case without overload. The difference for mechanical spring back is easy to perceive, and the reason of overloading is easy to understand.

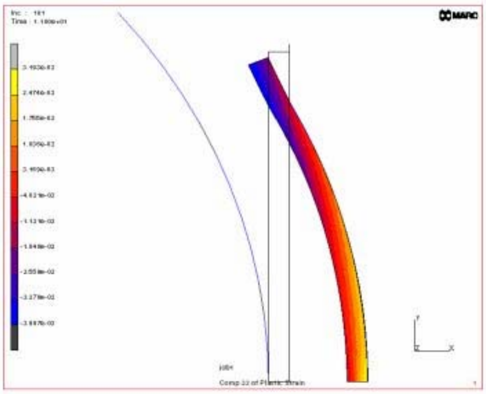

Figure 3. The bending simulation for the case with overload

As the constant dilatation and the assumed strain formulation for arbitrary quadrilateral axisymmetric four node isoparametric elements was utilised for the computation, it can be seen from the figure that the plastic region starts from outside of the sheet and spreads to inside area as it is seen in the elementary beam theory of elastic-plastic materials. It can be also identified the so called neutral line or in investigated case surface, where theoretically exists no deformations (elongation or compression), but that it not so obvious as in plane strain case. The reason for that is that in an axisymmetric case and in fact, three-dimensional state of stress there exist considerable tangential stresses. Although as it is shown on the Fig. 6 the distribution of the longitudinal stress components σ_{33} of the node 12 and 22 (nodes on the opposite sides of plate at the top of the sheet) corresponding to the last incremental stages of bending is not purely symmetric, the absolute values are almost the same thus assuring the minimum mechanical spring back of plate for that case.

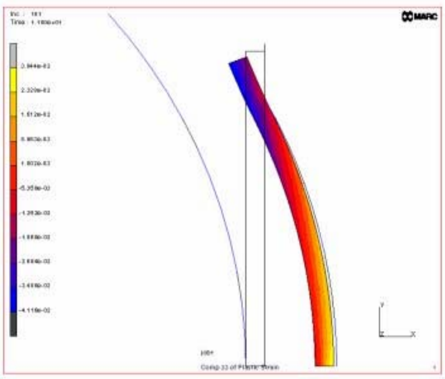

Figure 4. The bending simulation for the case without overload

Involving the friction force in the analysis of the bending process shows the somewhat higher stresses in the plate. It can be perceived some influence of the friction coefficient on the final shape of the plate. When the friction coefficient is taken zero value, the final shape, especially at the middle zone, looks shapely. However, the evaluation of the friction influence will be examined by the laboratory experiments.

The spring back effect is considered for the different bending conditions including an overloading and/or a friction. As a measure for mechanical spring back it is used the distance between the top of the bended plate and curvature of the die. As it can be seen from the Table 1. The spring back distance after an unloading is the smallest in the bending model, including friction and the overloading effect. Obviously, when the overloading is taking place the spring back should be considerably smaller since it is in fact the role of the overloading treatment.

Table 1.

State of friction and loading		Mechanical spring back as displacement in [mm]	
		Measured value	Computed value
Without overload	No friction	-	5.093
	With friction	3.245	3.347
With overload	No friction	-	1.526
	With friction	0.279	0.283

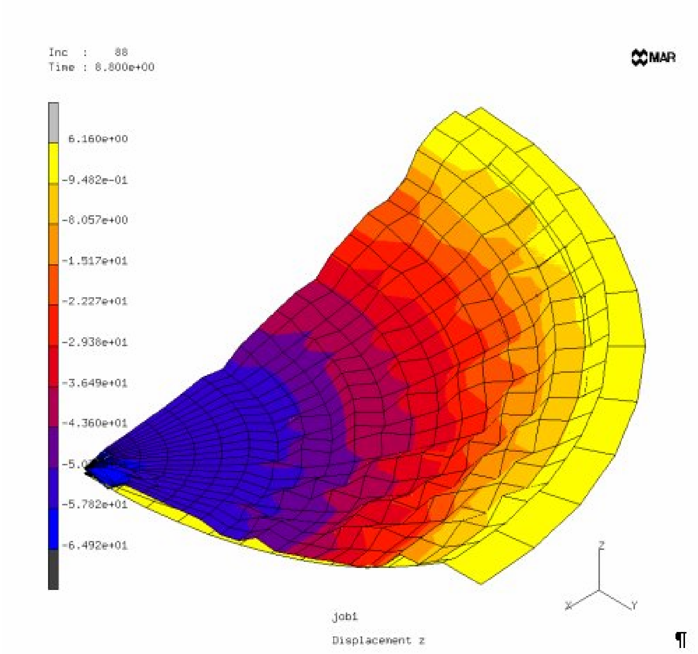

Figure 5. The simulation of bending using 3D membrane shell elements

5. DISCUSSION ON ACCURACY OF THE SOLUTION

The considered bending is analysed by the utilising the four node isoparametric quadrilateral elements as well as membrane 3D shell elements (see Fig. 5). As it was previously indicated the first one of these elements made possible the use of the assumed strain and constant dilatation that is very important when the large deformations is taking place, while the second ones allow bilinear thickness in the plane of the element. Very often the authors use the constant strain triangular elements, and these type was in additional analysis applied to verify the validity of the obtained results. Unfortunately, for the six node triangular element no constant dilatation option can be used in MARC, but only constant stress. It was found that the deformation is slightly different with this type of elements, but the mechanical spring back is not significantly different. Although, the stresses are notable different in all cases, it is obviously the consequence of the poor behaving in the bending model and there is no the possibility of the linear variation in the shear strain. Therefore, the picture of the distribution of the stresses is somewhat different when the triangular constant stress elements are used. Using 3D membrane shell elements it was possible to perceive the problem of wrinkling at the edges of the plate and that was avoided using overloading that was used at the same time for decreasing unavoidable mechanical spring back.

It is very important to conclude about the size i.e. the number of elements and nodes that are needed for correct description of the process. When just some of the elements include the adaptive meshing method, the deformation process does not correspond to the real one, and the measured values of mechanical spring back indicate considerably lower values then those given by the finite element analysis. In the example when the number of elements is large enough or the adaptive meshing includes all elements, the mechanical spring back computed by the analysis indicates no significant difference comparing to the real case.

It is also very important to conclude about the size of the increment, were it was found that this value has even greater influence on the mechanical spring back then the size of the single element, but not so great influence on the picture and distribution of the stresses and its values in detached nodes and overall. As it is also recommended in MARC it was find very useful to done the separation in very small increments, because a solution requiring artificial stretching might have to be found before the decision to separate is made.

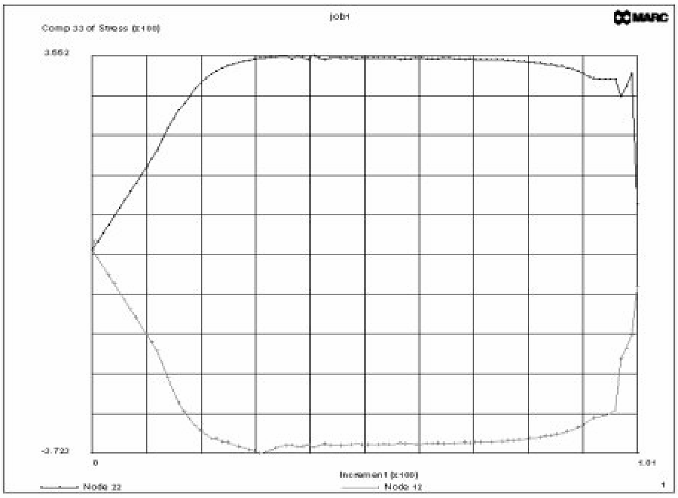

Figure 6. Distribution of $\sigma_{33\,for}$ nodes 12 and 22

Two possible FE models of bended plate are discussed. In the first one four-node isoparametric quadrilateral elements were used applying both punch motion option as well as pressure table option. In both cases obtained mechanical spring back is in good correspondence with experimental data. In second FE model membrane 3D shell elements were used enabling to get more informant picture of the process. In addition, the load-punch motion curve is constructed using relevant subroutine that determines the total load placed on the structure for each increment. The load punch motion curve for this problem is calculated from the total reaction forces in the plane of symmetry. The total reaction force on the tool interface is the same. As it can be seen from the Fig. 7. For the case with overload, characteristic of this curve correspond in good approximation with the experimental ones. Using FE approach it is possible to simulate the bending process and unavoidable mechanical spring back, thus improving the analysis of process conditions and die design are made at the engineering stage before dies are manufactured and expensive production machinery is tied up for costly experimentation. It is very important to conclude about the number of elements i.e. number of nodes as well as number and amount of increments, which divide the whole bending process into small computation steps. Of course, at the beginning of the investigation it is recommended to use less elements and more crude increments, but the later phases of investigation needed more elements and finer increments to be taken into calculation.

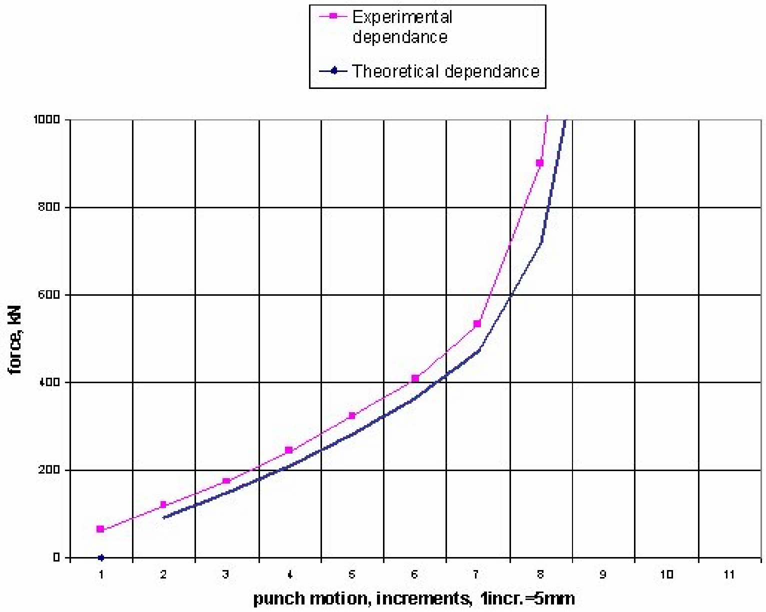

Figure 7. The load-deflection curve for measured and computed cases with overloading

6. CONCLUSION

With FEM it has become possible to predict both the magnitude and the distribution of the stresses in a work piece due to the forming and thus it has become possible to optimise the forming process with regard to the stresses, form and number of stages as well as the needed forces and work of deformation.

The residual stress calculation indicates that the solution is somewhat in equilibrium. Compared to the reaction forces, the errors in nodal equilibrium are in order of 1%. The equivalent plastic strain is calculated from the strain components. The quality of the mesh proved to be an essential factor in performing successfully FEM analysis. The number, size and shape of the elements are of importance for the solution accuracy. The number of nodes influences especially regarding the simulation of friction forces. The possibility of analysis the local stress-strain rate and stress distribution enhance an improved tool design, prevention of premature die failure and elastic displacement of the tooling might assist in the process layout phase, where a corrected tool geometry can be developed. The results showed here, demonstrate the final element approach and simulation as a useful technique in studding the process of plate bending, where there is a generally a close correlation in the load results obtained with FEM and the experimental ones. The numerical modelling and simulation offer reliable method for exploration the metal forming technologies besides the problems with contact and friction stresses and conditions mentioned above. The correctness of the computed results is still dependent on the selection made regarding various modelling parameters. Among the most important aspects, which can be mentioned, the constitutive law and the boundary conditions as well as correct mesh and type of elements play a decisive role in achievement of correct results. As a general rule, computational costs are rising with the desired precision, reliability and quality of results.

7. REFERENCES

[1] ALTAN, T., - Application of CAD/CAM in precision forming, Steel research, VDI, Nr. 5/88, 1988, pp. 212-219

[2] MALGYN D., KARDOS K., KRALLICS GY. - 3D Simulation of square cup drawings, Advanced Technology of Plasticity, Volume 2,1999, pp. 661-662, Springer Verlag

[3] LANGE, K., - Lehrbuch der Umformtechnik, Springer-Verlag, Berlin, Heidelberg, New York, 1993

[4] DOEGE, E., MEYER-NOLKEMPER. H., SAEED, I., - Fließkurvenatlas metalischer Werkstoffe, Hanser-Verlag, München-Wien, 1986

[5] DEJMAL I., TIROSH J., SHIRZLY A., - The effect of die curvature on the punch drawing force in deep drawing processes, Advances in Manufacturing Technology, pp. 53-59, University of Bath, 15 th National Conference on Manufacturing Research, United Kingdom, 1999.

[6] ONATE E., - New degrees of freedom in computational mechanics: mesh free finite point method, rotation free shell triangles and moving meshes, NAFEMS World Congress, Newport, Rhode Island, USA, 1999, pp 1-27

[7] DUTILLY M., GELIN J.C., Design of sheet metal forming processes based on quality functions, ESAFORM Conference, Guimaraes, Portugal, pp 469-472, 1999

Assoc. Prof. Miljenko Math, Ph.D., University of Zagreb, Faculty of Mechanical Engineering & Naval Architecture, Department of Technology, Chief of Laboratory for Metalworking, 10000 Zagreb, Ivana Lučića 5, tel.: (01) 61 68 222, fax. (01) 61 56 940, e-mail: miljenko@fsb.hr, C R O A T I A

Ass. Zdenka Keran, B.Sc., Young researcher, University of Zagreb, Faculty of Mechanical Engineering & Naval Architecture, Department of Technology, Zagreb, Ivana Lučića 5, tel.: (01) 61 68 222, fax.:(01) 61 56 940, e-mail: zdenka@fsb.hr, C R O A T I A

Asc. Prof. Branko GRIZELJ, Ph.D., Faculty of Mechanical Engineering in Slavonski Brod, University in Osijek, 35000 Slavonski Brod, e-mail: grizelj@sfsb.hr, C R O A T I A

ANALIZA PROCESA SAVIJANJA PLOČA POMOĆU METODE KONAČNIH ELEMENATA

M. Math, Z. Keran, B. Grizelj

REZIME

Savijanje lima je standardna operacija u mnogim industrijskim procesima. Uprkos činjenici da je ova tehnologija široko rasprostranjena, konstrukcije alata i izbor materijala lima je još uvek uglavnom baziran na eksperimentalnim osnovama (trial and error).

Mehanika kontinuuma je veoma kompleksna da bi se pomoću nje mogla dobiti egzaktna, konkretna rešenja. U takvoj situaciji primena numeričkih metoda, pre svega metoda konačnih elemenata pokazala se kao veoma moćno sredstvo u rešavanju konkretnih problema u oblasti obrade deformisanjem.

Pomoću ove metode moguće je odrediti veličinu i raspored napona u obratku a time se omogućava optimizacija samog procesa u odnosu na napone, oblik i broj operacija, veličinu deformacione sile i rada.

U ovom radu prikazana je analiza procesa savijanja ploča pomoću metode konačni elemenata. Dobijeni rezultati te analize upoređeni su sa eksperimentalnim rezultatima. Analiziran je komad-deo sferičnog rezervoara koji se dobija na hidrauličnoj presi.

Kvalitet mreže za FEM je jedan od ključnih faktora za krajnji rezultat. Broj tačaka mreže posebno utiče na simulaciju fenomena kontaktnog trenja a broj, oblik i veličina elemenata mreže znatno utiču na tačnost dobijenih rezultata.

Rezultati koji su prikazani u ovom radu pokazuju da je FE metoda veoma korisna za analizu procesa savijanja ploča. Tačnost dobijenih rezultata, dakako zavisi i od izbora različitih parametara modelovanja. Najvažniji od njih su svakako granični uslovi, konstruktivni zahtevi, karakteristika mreže. Generalno, troškovi kompjuterske obrade rastu sa porastom zahtevane preciznosti, pouzdanosti i kvantitetom željenih rezultata.