IMPROVEMENT OF STABILITY CONDITIONS AND UNIQUENESS OF PENALTY APPROACH IN CONTACT MODELLING IN SHEET METAL FORMING

***Tomasz Bednarek*[1,2], *Piotr Kowalczyk*[1], *Artur Marczewski*[1], *Włodzimierz Sosnowski*[1,2]**
[1]*Institute of Fundamental Technological Research of the Polish Academy of Sciences, 02-106 Warsaw, Pawinskiego 5B, Poland*
[2]*Kazimierz Wielki University, 85-064 Bydgoszcz, Chodkiewicza 30, Poland*

Abstract

The main objective of this paper is to improve stability conditions, uniqueness and convergence of the flow approach algorithm with viscoplastic and plastic material models. In this paper, the problem of convergence and uniqueness of the problem of non-linear simulation of sheet metal forming processes modeled using rigid-viscoplastic material model is considered. In the numerical simulation of the deformation process MFP2D and MFP3D Finite Element programs were used. The simplicity of the algorithm is the main advantage of these codes, the Direct Differentiation method and optimization modules can be implemented in the source code. The numerical instability caused by high values of the condition number of the main system of equations is the main disadvantage of the codes. The penalty approach contact model used in the program makes the stiffness matrix condition number worse.

1. Introduction

The objective of this paper is to improve stability conditions, uniqueness and convergence of the flow approach algorithm with viscoplastic and plastic material models. The numerical codes MFP2D and MFP3D are used for practical simulations of the industrial processes. In this paper authors focus on the contact modeling by modifications of the penalty approach method.

In the previous authors' paper (Sosnowski *et al.*, 2010), the stability and uniqueness of flow approach algorithm in sheet metal forming simulations were studied. In the present paper, new results in this field are presented.

One of significant drawbacks of rigid-viscoplastic shell approach is poor stability and lack of convergence due to relatively high values of the condition number of the main system of equations. The main reason that brings out the relatively bad condition of the system of equations is the penalty approach treatment of the contact modeling. The authors' previous study on metal forming processes proves that the choice of the proper value of the penalty factor is one of the main problems in computations. A too small value of the penalty factor causes bad accuracy of the contact modeling. Assumption of a too large value of penalty factor leads to poor system conditioning and significant errors. In many cases, i.e. when the geometry of the drawpiece is complex, the satisfactory value of penalty factor may not exist at all. In many commercial and academic programs the penalty factor is assumed constant during the whole forming process and selected arbitrary, according to the engineer's experience at the moment.

The authors propose a modification of the well known penalty approach in contact modeling. The main aim is to find a relationship which allows to calculate the best value of the penalty factor for given drawing conditions. It should be noticed that the penalty factor does not need to be constant for all nodes. Additionally, the penalty factor does not need to be constant for all iterations or time steps. In the proposed approach, only the limit value of the tool penetration in sheet have to be assumed instead of choosing abstract value of penalty factor. On the basis of the material parameters, geometry and actual configuration of the sheet and the acceptable penetration value, the penalty factor is calculated separately for each node of the sheet in contact with tool at all time or iteration steps. At result, the precision of the contact modeling is approximately the assumed limit value of the penetration.

The tests performed show very good and promising results. Especially the influence of proposed approach on the condition number of the main system of equations can be noticed. Authors observe really good effect in the case of the contact modeling. The condition number of the system of equations was improved at least by two orders of magnitude in relation to the system of equation without any contact conditions.

2. Flow approach formulation in sheet metal forming

The flow approach to metal forming problems with the rigid-viscoplastic material model is used as the basis in this paper (Perzyna, 1966; Oñate and Agelet, 1992).

The virtual work expression (equilibrium equation in the weak form) to be solved reads

$$\int_{\Omega} \sigma_{ij} \delta \dot{\varepsilon}_{ij} d\Omega = \int_{\Omega} f_i \delta v_i d\Omega + \int_{\partial\Omega_t} t_i \delta v_i d(\partial\Omega), \qquad i, j = 1, \ldots, 3 \tag{1}$$

where v_i denotes velocity field, f_i is the distributed volumetric load, t_i is the traction on the boundary an integrals are taken over the actual body volume element $d\Omega$ or its surface element $d(\partial\Omega)$, respectively.

Strain rates are presented as

$$\dot{\varepsilon}_{ij} = \frac{1}{2} \left(v_{i,j} + v_{j,i} \right) \tag{2}$$

Stresses are calculated from the constitutive equation

$$\sigma_{ij} = s_{ij} + p\delta_{ij} \,, \qquad s_{ij} = 2\mu^* \dot{\varepsilon}_{ij} \tag{3}$$

where s_{ij} is the Cauchy stress deviator, p denotes the mean stress and δ_{ij} is the Kronecker delta.

The constitutive function μ^* is defined in the flow problem as

$$\mu^* = \frac{\overline{\sigma}}{3\dot{\overline{\varepsilon}}} = \frac{1}{3\dot{\overline{\varepsilon}}} \left[\sigma_y + \left(\frac{\dot{\overline{\varepsilon}}}{\gamma} \right)^{\frac{1}{n}} \right] \tag{4}$$

Here, σ_y is the current static uniaxial tensile yield stress of the material, $\overline{\sigma} = \sqrt{\frac{3}{2} s_{ij} s_{ij}}$ is the equivalent stress, $\dot{\overline{\varepsilon}} = \sqrt{\frac{2}{3} \dot{\varepsilon}_{ij} \dot{\varepsilon}_{ij}}$ is the effective inelastic strain rate, and γ, n are physical parameters of the rigid-viscoplastic model used.

For plastic materials with strain hardening, the yield limit $\sigma_y \left(\overline{\varepsilon} \right)$ is a function of the effective inelastic strain $\overline{\varepsilon}$, and $\overline{\varepsilon}$ has to be computed as the time integral of $\dot{\overline{\varepsilon}}$.

The formal analogy between the plastic flow equations and a formulation of incompressible elasticity allows to solve the pure plastic flow problem with a numerical code developed for linear elasticity. Rates of large plastic strains are treated in the same way as elastic strains. The incompressibility condition must only be satisfied.

In sheet metal forming, the shell theory is used as the simplification of 3D problems. Plane stress assumptions are used in shell theory so that the incompressibility can be easily achieved by adjusting the shell thickness during consecutive steps of the solution to ensure the constant volume.

After spatial finite element discretization the stiffness matrix $\mathbf{K}$ depends on the nodal velocities $\dot{\mathbf{q}}$ through the parameter μ^* so that an iterative process is needed to find the solution vector $\dot{\mathbf{q}}$.

$$\mathbf{K}^{(k)}\left[\mu^*\left(\dot{\mathbf{q}}^{(k)}\right)\right]\dot{\mathbf{q}}^{(k+1)} = \mathbf{Q} \qquad k = 0, 1, 2, \ldots \tag{5}$$

in which $\mathbf{Q}$ denotes the external force vector, and k is the iteration number.

3. Penalty function method

The crucial factor in finite element modelling of realistic technological metal forming problems is the way the contact between sheet and rigid bodies is treated.

In previous studies (Sosnowski *et al.*, 2010), an algorithm for computing the ratio of the main determinants of the system of equations, stiffness matrix, was developed and implemented in metal forming simulation software MFP2D. One of the main reasons of the bad conditioning of the problem was the penalty function method implemented in the contact computations. An alternative way of modelling the contact, the Lagrangian multipliers method, was also considered. The main disadvantage of this method is that the main root system of equations is extended by an additional multipliers. Their number corresponds to the number of degrees of freedom of the system where restrictions resulting from the contact are imposed. An additional difficulty is the fact that the number of multipliers varies not only in different time steps but also on the given time step in subsequent iterations. Because of this drawbacks in the current state of research Lagrangian multipliers method was considered too expensive numerically, as well as causing potential difficulties due to the variable size of the problem, for effective implementation in metal forming MFP2D and MFP3D simulation codes.

Functional of the potential energy is supplemented by a part that represents the potential energy of contact constraints

$$\Pi(\mathbf{u}) = \frac{1}{2}K_{ij}u_iu_j - Q_iu_i + \epsilon\frac{1}{2}(D_{ki}u_i - \hat{u}_k)(D_{kj}u_j - \hat{u}_k), \tag{6}$$

where D_{ki} is a Boolean matrix identifying an active contact-restricted degrees of freedom while $\hat{u}_k$ is a vector of contact constraints. The condition of stationarity of the first derivative has the form

$$\frac{\partial \Pi}{\partial u_i} = K_{ij}u_j + \epsilon D_{ik}D_{kj}u_j - \epsilon D_{ik}\hat{u}_k - Q_i = 0 \tag{7}$$

After some transformations we have

$$\left(K_{ij} + \epsilon D_{ik}D_{kj}\right)u_j = Q_i + \epsilon D_{ik}\hat{u}_k \tag{8}$$

The penalty function method requires no expansion of system of equations with additional unknowns. Regardless of the number of restrictions the size of the problem remains the same. However, serious difficulties arise from the necessity to determine the penalty factor of ϵ. A too low value of the constant penalty factor ϵ decreases the accuracy of determining the contact. On the other hand the large value of ϵ affects the matrix conditioning. In the case of metal forming nonlinear simulations it is very difficult to determine the proper value of the penalty factor coefficient. It is determined by the trial and error method which is inefficient in the industry metal forming simulations.

4. Main modifications of the penalty method

In this section, the modification of penalty approach method is proposed. The main idea is to apply the accuracy coefficient (i.e. the absolute error or the depth of penetration) instead of fixing the penalty coefficient during the whole simulation of the forming process.

The penalty function method requires the adoption of some value of the penalty factor ϵ. It is not obvious what should be the value of this factor. The value of the penalty factor should be chosen as a large number. Hovewer, a too large value of the factor causes significant deterioration in matrix condition number.

The penalty factor could be associated with artificially introduced rigidity to a particular element of the stiffness matrix of the problem. The solution of the system of equations must of course satisfy imposed restrictions. In addition, the rightr- hand side of the system of equations is modified.

The value of the penalty factor coefficient does not need to be the same for each degree of freedom. In addition, the penalty coefficient may vary in subsequent time steps and in subsequent iterations during the solution of nonlinear equation problem. The only assumption is made that on the left and right side of system of equations the penalty coefficients values for a given degree of freedom have to be equal. This assumption results from derivation of the the basic system of equations (8).

The main idea of modifications of the penalty method algorithm in the contact problem is to improve the accuracy (i.e. the absolute error or the depth of penetration) instead of fixing the penalty coefficient value (the artificial rigidity) during the whole metal forming simulations.

4.1. Modifications of the algorithm for one degree of freedom

The model shown in Figure 1 is considered; k is a stiffness of the spring, q is the exciting force, $\hat{u}$ is the assumed value of displacement (restriction resulting from the contact), ϵ is the penalty factor and δ is the limit depth of penetration.

The potential energy of the system presented in Figure 1 is given as

$$\Pi = \frac{1}{2}ku^2 - qu + \frac{1}{2}\epsilon(\hat{u} - u)^2 = \frac{1}{2}ku^2 - qu + \frac{1}{2}\epsilon\hat{u}^2 - \epsilon\hat{u}u + \frac{1}{2}\epsilon u^2. \tag{9}$$

From the condition of stationarity of the first derivative of the potential energy we have

$$ku - q - \epsilon\hat{u} + \epsilon u = 0. \tag{10}$$

After some transformations we come at

$$(k + \epsilon)u = q + \epsilon\hat{u}. \tag{11}$$

The displacement u is thus given as

$$u = \frac{q + \epsilon\hat{u}}{k + \epsilon} \tag{12}$$

For large values of ϵ, the value of displacement u is close to the value of $\hat{u}$.

Permitted penetration depth δ takes the form

$$\hat{u} - u \leq \delta \tag{13}$$

Increase in the value of the penalty factor ϵ causes decrease in the value of the coefficient δ given by equation (13). Assuming that the given value of the penetration is satisfactory the inequality (13) could be replaced by the equality

$$\hat{u} - u = \delta. \tag{14}$$

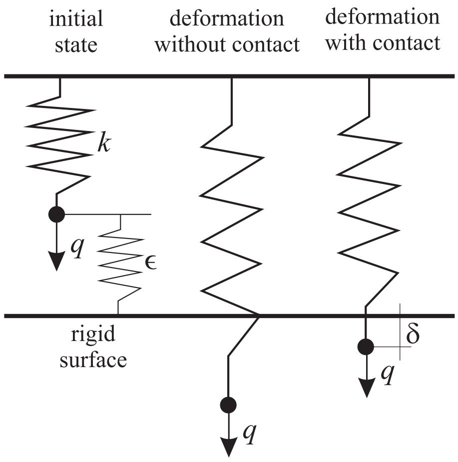

Figure 1. Elastic spring with contact constraint

Substituting the relation (12) to equation (14) we get

$$\hat{u} - \frac{q + \epsilon \hat{u}}{k + \epsilon} = \delta. \tag{15}$$

After some additional transformations the equation takes the form

$$\epsilon = \frac{k\left(\hat{u} - \delta\right) - q}{\delta}\ . \tag{16}$$

The above equation allows to determine the penalty factor coefficient in the way that displacement is calculated with the assumed accuracy δ.

4.2. Formulation for the two-dimensional case

As in the case of one-dimensional problem, the basic system of N equations is derived from the minimum of the potential energy condition

$$\mathbf{K}^{*}\mathbf{u} = \mathbf{q}^{*} \tag{17}$$

where $\mathbf{u}$ is the solution vector of the system — the nodal displacements or velocitiwes, while * denotes arrays modified due to displacement restrictions,

$$\mathbf{K}^{*} = \mathbf{K} + \boldsymbol{\epsilon}, \quad \mathbf{q}^{*} = \mathbf{q} + \boldsymbol{\epsilon}\left(\mathbf{D}\hat{\mathbf{u}}\right). \tag{18}$$

Here, $\boldsymbol{\epsilon}_{N\times N}$ is the penalty coefficient matrix, $\mathbf{D}_{N\times m}$ is the Boolean matrix identifying the degrees of freedom with contact displacement restrictions, $\hat{\mathbf{u}}_m$ is the vector of the contact restriction values, N is the total number of degrees of freedom and m is the total number of contact restrictions.

We define the operator $|\mathbf{a}|$ resulting in the vector of the absolute value of the elements $\mathbf{a}$

$$|\mathbf{a}| = |a_i|_i, \quad |\mathbf{a} - \mathbf{b}| = |a_i - b_i|_i \tag{19}$$

We assume the accuracy of modeling of the contact δ. In the case of m restrictions, the accuracy of modeling is expressed by the vector

$$\boldsymbol{\delta} = \delta\mathbf{1} \tag{20}$$

where $\mathbf{1}$ is the m-dimensional unit vector. The equations take the form

$$|\mathbf{D}\hat{\mathbf{u}} - \mathbf{u}| = \mathbf{D}\boldsymbol{\delta}. \tag{21}$$

Substituting the solution of the global system of equations (17) to equations (21) we get

$$\left|\mathbf{D}\hat{\mathbf{u}} - (\mathbf{K}+\boldsymbol{\epsilon})^{-1}\left(\mathbf{q}+\boldsymbol{\epsilon}\left(\mathbf{D}\hat{\mathbf{u}}\right)\right)\right| = \mathbf{D}\boldsymbol{\delta} \tag{22}$$

Multiplying both sides by $(\mathbf{K}+\boldsymbol{\epsilon})$ we get

$$|\mathbf{K}\mathbf{D}\hat{\mathbf{u}} + \boldsymbol{\epsilon}\mathbf{D}\hat{\mathbf{u}} - \mathbf{q} - \boldsymbol{\epsilon}\mathbf{D}\hat{\mathbf{u}}| = \mathbf{K}\mathbf{D}\boldsymbol{\delta} + \boldsymbol{\epsilon}\mathbf{D}\boldsymbol{\delta}. \tag{23}$$

After transformation of equation (23) we get

$$.\boldsymbol{\epsilon}\mathbf{D}\boldsymbol{\delta} = |\mathbf{K}\mathbf{D}\hat{\mathbf{u}} - \mathbf{q}| - \mathbf{K}\mathbf{D}\boldsymbol{\delta} \tag{24}$$

After sidewise left-multiplication by $\mathbf{D}^T$ we obtain the reduced system of equations

$$\overline{\boldsymbol{\epsilon}}\boldsymbol{\delta} = \left|\overline{\mathbf{K}}\hat{\mathbf{u}} - \overline{\mathbf{q}}\right| - \overline{\mathbf{K}}\boldsymbol{\delta} \tag{25}$$

where

$$\overline{\mathbf{K}}_{m\times m} = \mathbf{D}^T\mathbf{K}\mathbf{D}, \qquad \overline{\boldsymbol{\epsilon}}_{m\times m} = \mathbf{D}^T\boldsymbol{\epsilon}\mathbf{D}, \qquad \overline{\mathbf{q}}_{m\times 1} = \mathbf{D}^T\mathbf{q} \tag{26}$$

In equation (25), the right-hand side and $\boldsymbol{\delta}$ are vectors while $\overline{\boldsymbol{\epsilon}}$ is the unknown matrix of penalty coefficients. The eigenvalues of the matrix $\overline{\boldsymbol{\epsilon}}$ are calculated because there is no possibility of calculating the matrix full form. The matrix of the penalty coefficients $\overline{\boldsymbol{\epsilon}}$ takes the diagonal form

$$\overline{\epsilon}_{ij} = \begin{cases} 0 & \text{for} \quad i \neq j \\ \frac{\left|\overline{K}_{ik}\hat{u}_k - q_i\right| - \overline{K}_{ik}\delta_k}{\delta_i} & \text{for} \quad i = j \end{cases} \tag{27}$$

The penalty coefficients in equation (27) are transformed to global form

$$\boldsymbol{\epsilon} = \mathbf{D}\overline{\boldsymbol{\epsilon}}\mathbf{D}^T. \tag{28}$$

To illustrate the formulation, consider the unilaterally restrained beam shown in Figure 2. The beam is discretized using three finite elements. The cross-section is a 1×1 mm square while the beam length l is 30 mm. The stiffness matrix of a single finite element is given by

$$k^e = \begin{bmatrix} \frac{EA}{l} & 0 & 0 & -\frac{EA}{l} & 0 & 0 \\ & \frac{12EJ}{l^3} & \frac{6EJ}{l^2} & 0 & -\frac{12EJ}{l^3} & \frac{6EJ}{l^2} \\ & & \frac{4EJ}{l} & 0 & -\frac{6EJ}{l^2} & \frac{2EJ}{l} \\ & & & \frac{EA}{l} & 0 & 0 \\ & & symm & & \frac{12EJ}{l^3} & -\frac{6EJ}{l^2} \\ & & & & & \frac{4EJ}{l} \end{bmatrix} \tag{29}$$

where A is the cross-section area, J is the moment of inertia and $E = 210\ GPa$ is the Young modulus. The accurate value of the contact modelling is assumed as 0.1 mm. The global stiffness is assembled from element matrices and reads (we skip units)

$$K = \begin{bmatrix} \mathbf{k}^1_{[4,4-6,6]} + \mathbf{k}^2_{[1,1-3,3]} & \mathbf{k}^2_{[1,3-3,6]} & \mathbf{0} \\ & \mathbf{k}^2_{[4,4-6,6]} + \mathbf{k}^3_{[1,1-3,3]} & \mathbf{k}^3_{[1,3-3,6]} \\ symm & & \mathbf{k}^3_{[4,4-6,6]} \end{bmatrix}$$

$$= \begin{bmatrix} 420 & 0 & 0 & -210 & 0 & 0 & 0 & 0 & 0 \\ & 4.2 & 0 & 0 & -2.1 & 10.5 & 0 & 0 & 0 \\ & & 140 & 0 & -10.5 & 35 & 0 & 0 & 0 \\ & & & 420 & 0 & 0 & -210 & 0 & 0 \\ & & & & 4.2 & 0 & 0 & -2.1 & 10.5 \\ & & & & & 140 & 0 & -10.5 & 35 \\ & & & & & & 210 & 0 & 0 \\ & & symm & & & & & 2.1 & -10.5 \\ & & & & & & & & 70 \end{bmatrix} \tag{30}$$

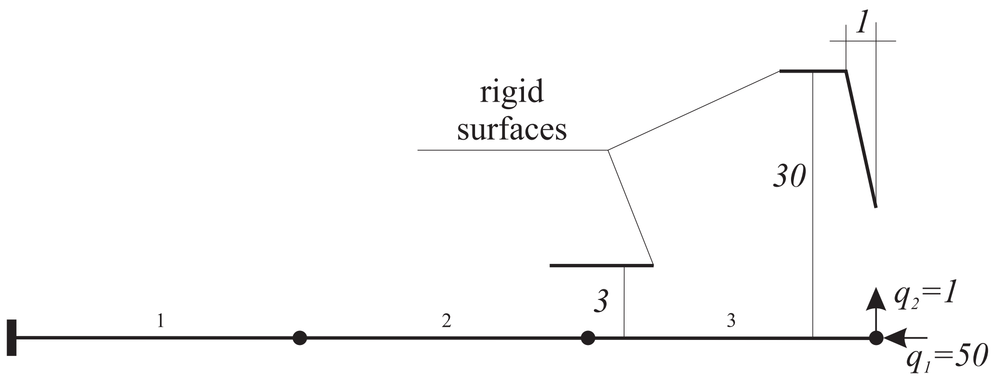

Figure 2. Restrained beam

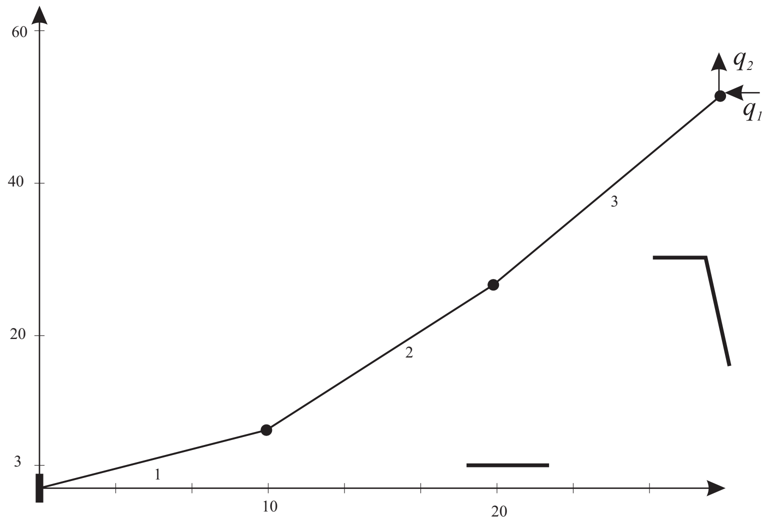

Figure 3. The beam deflection without contact conditions

The determinant of the stiffness matrix is $|\mathbf{K}| = 4.597e11$. The matrix condition number is $uw(\mathbf{K}) = 4.572e4$.

The load vector (see Figure 2) has the following components

$$\mathbf{q} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & -50 & 1 & 0 \end{bmatrix}^T \tag{31}$$

The displacement vector computed as if the contact conditions were disregarded ($\mathbf{K}\mathbf{u}_{bk} = \mathbf{q}$) is

$$\mathbf{u}_{bk} = \begin{bmatrix} -0.238 & 7.619 & 1.429 & -0.476 & 26.667 & 2.286 & -0.414 & 51.429 & 2.571 \end{bmatrix}^T \tag{32}$$

The beam deflections for this case are shown in Figure 3.

Regarding contact constraints, rigid surfaces limit the transverse displacement of the node 3 and both the displacement components of node 4 (see Figure 2). The vector of contact constraints and the identification table have the form

$$\hat{\mathbf{u}} = \begin{bmatrix} 3 \\ -1 \\ 30 \end{bmatrix}, \qquad \mathbf{D}^T = \begin{bmatrix} 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix} \tag{33}$$

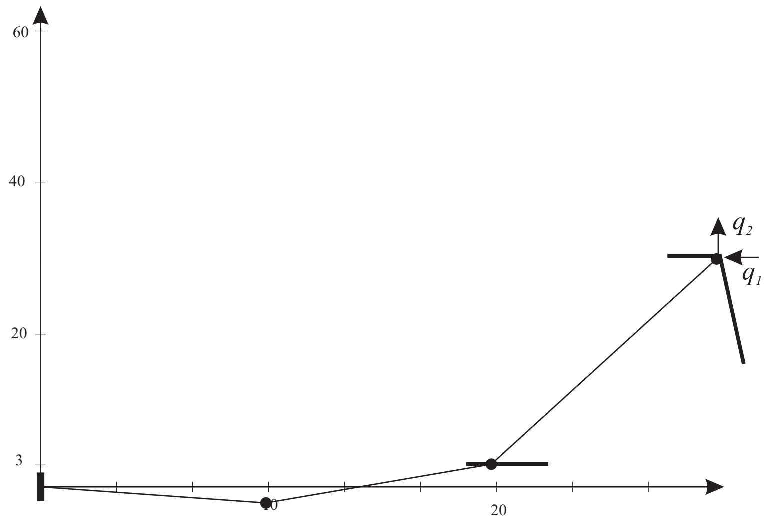

Figure 4. The beam deflection with contact conditions present

The matrix of the penalty coefficients calculated with equations (27) and (28) has the form

$$\boldsymbol{\epsilon} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 501.9 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1390 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 557 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \quad (34)$$

The determinant of the stiffness matrix modified by the contact condition set of equations is $|\mathbf{K} + \boldsymbol{\epsilon}| = 1.948e20$. The matrix condition number is $uw(\mathbf{K} + \boldsymbol{\epsilon}) = 513.871$. The decrease of the matrix condition number by two orders of magnitude was observed ($uw(\mathbf{K}) = 4.572e4$). Modifications of boundary conditions by moving the rigid surfaces position results in possible increase of the matrix condition number. However, even the worst result observed were still better than $5.0e3$. The vector of the nodal displacements takes the following values

$$\mathbf{u} = \begin{bmatrix} -0.329 & -2.764 & -0.201 & -0.658 & 3.018 & 1.709 & -0.986 & 29.992 & 3.192 \end{bmatrix}^T \quad (35)$$

The beam deflection is shown in Figure 4.

The difference between coordinates of the rigid surfaces and the position of the beam node after deformation is

$$\mathbf{D}^T\mathbf{u} - \hat{\mathbf{u}} = \begin{bmatrix} 0.018 \\ 0.014 \\ 0.007 \end{bmatrix} \text{ [mm]} \quad (36)$$

It should be noted that the above difference is an order of magnitude smaller than assumed $\boldsymbol{\delta}$. This is probably due to the fact that there is no possibility of calculating the full matrix of the penalty coefficients $\overline{\boldsymbol{\epsilon}}$ (see equation (27)). However the distance between the rigid surface and beam element is always smaller then the assumed allowed penetration level. The change in the boundary conditions, ie. the position of the rigid surface or the value of the load, resulted in

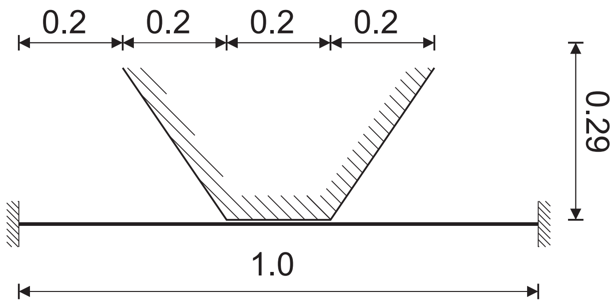

Figure 5. Test example, dimensions are given in inches

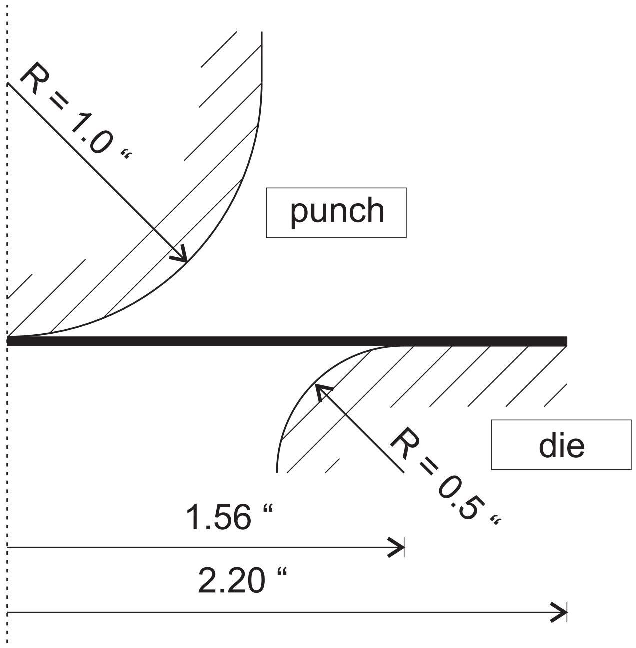

Figure 6. Hemispherical punch deep drawing problem, dimensions are given in Inches

exceeding the projected value of δ only in extreme cases, such as the value of displacements without contact was a few rows greater than the contact limits $\hat{\mathbf{u}}$. Such cases seldom occur in real numerical computations.

5. Numerical examples

The first numerical example considered is the test example with geometry shown in Figure 5. The initial thickness of the blank was equal 0.035 in. The Coulomb friction coefficient was assumed 0.2.

There were two cases studied. One with classical penalty approach with penalty coefficient 1.0E+07 and with modified approach with accuracy assumed 1.0E-05. The improvement in matrix condition number was observed with the use of modified approach. The matrix condition number in modified approach was equal 3.721074E+02 compared to 2.705008E+05 obtained with the usage of classical penalty approach method.

Deep drawing of a thin circular isotropic sheet with a hemispherical punch is considered as a second example (Woo, 1968). The geometrical configuration of the problem is shown in Figure 6. The initial radius of the blank is 2.22 in. The initial thickness of the blank was equal 0.035 in. The Coulomb friction coefficient was 0.2.

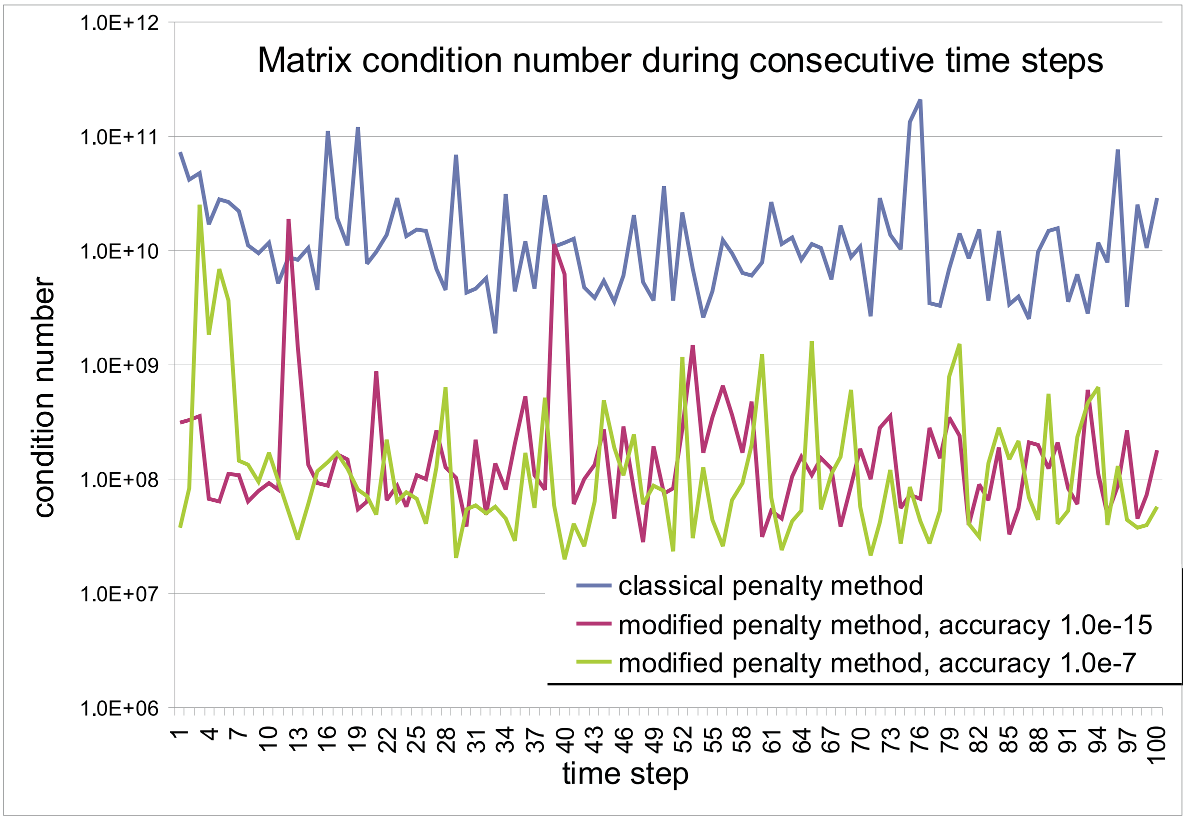

Figure 7. Matrix condition number during consecutive steps

Fifty linear bending elements have been used in the analysis. The analysis was performed with the use of MFP2D program with axisymmetric algorithm option. The strain hardening law was assumed in the form (stress unit is [ton/in^2])

$$\overline{\sigma} = \begin{cases} 5.4 + 27.8\overline{\varepsilon}^{0.504}, & \text{for } \overline{\varepsilon} \leq 0.36 \\ 5.4 + 24.4\overline{\varepsilon}^{0.504}, & \text{for } \overline{\varepsilon} > 0.36 \end{cases} \tag{37}$$

Summary indicators of determinants of matrices in all time steps of numerical simulations are shown in Figure 7. It should be noted that in all but one increments the matrix condition number is improved with the use of modified approach compared to classical penalty method approach.

Acknowledgements

This work was partially financed by European Regional Development Fund within the framework of the Innovative Economy Programme, project number POIG.01.03.01-14-209/09.

References

[1] Oñate, E.; Agelet de Saracibar, C. 1992: Numerical modelling of sheet metal forming problems. In: P. Hartley *et al.* (Eds.) *Numerical Modelling of Material Deformation Process*. Springer

[2] Perzyna, P. 1966: Fundamental problems in viscoplasticity. *Advances in Applied Mechanics* **9**, 243–377

[3] Sosnowski, W.; Bednarek, T.; Kowalczyk, P. 2010: Stability and uniqueness of flow approach algorithm in sheet metal forming simulations. *Computer Methods in Material Science* **10**(1), 30–36

[4] Woo D.M., 1968: On the complete solution of the deep drawing problem. *International Journal of Mechanical Sciences* **10**, 83–94

POPRAWA WARUNKÓW STABILNOŚCI I JEDNOZNACZNOŚCI DLA ZAGADNENIA KONTAKTOWEGO W ANALIZIE TŁOCZENIA BLACH

Streszczenie

Celem niniejszej pracy jest poprawa warunków stabilności, jednoznaczności i zbieżność zadania lepkoplastycznego płynięcia materiału tłoczonej blachy poprzez modyfikację procedury kontaktu z zastosowaniem zmiennego współczynnika kary. W niniejszej pracy rozważany jest problem zbieżności i jednoznaczności rozwiązania nieliniowego problemu symulacji procesów tłoczenia blach modelowanych z wykorzystaniem sztywno-lepkoplastycznego modelu materiałowego. W symulacji numerycznej procesu deformacji zastosowano programy MFP2D i MFP3D Metody Elementów Skończonych. Zaletą kodów numerycznych MFP2D i MFP3D jest prostota zastosowanego algorytmu analizy pozwalająca w konsekwencji na implementację modułów analizy wrażliwości metodami ścisłymi i optymalizacji procesu tłoczenia blach. Wadą przyjętego modelu materiałowego jest niestabilność numeryczna zadania wywołana wysokim wskaźnikiem uwarunkowania układu równań. Dodatkowo model kontaktu zawierający funkcję kary pogarsza wskaźnik uwarunkowania macierzy sztywności.