Thermally and Mechanically Coupled Simulation of Metal Forming Processes

S. Stebunov [1)*], N. Biba [1)], A. Vlasov [2)], A. Maximov [1)]

[1)] QuantorForm, Moscow / Russia; [2)] Moscow State Industrial University, Moscow / Russia

* Corresponding author: E-mail address: info@qform3d.com; Tel.: +7 495 663 33 80

Abstract

The paper presents the experience of development and implementation of the methods for simulation of the metal forming processes where the material flow problem is coupled with thermal and mechanical solutions in the tool set. The numerical model is built in the most wide ranging way that covers the needs of any kind of technological processes. Particularly, the deformable dies can be assembled and pre-stressed to provide favorable stress distribution under the load. The deformed workpiece itself may consist of several pieces of different materials having different mechanical and thermal properties and bound together. The workpiece material model may include thermal and elastic deformation components to provide the analysis of residual stresses in the finish product.

The model is realized in a new environment of version 7.0 of QForm2D/3D software.

Keywords: Forging, Simulation, Finite element

Introduction

The simulation of hot metal forming processes in most cases can be done using the assumption that the deformed material is a rigid-plastic continuum while the dies are rigid bodies (Biba, 2002). Nevertheless there are many cases where such simplification may result in inadequate accuracy. For example, the die deformation in cold forging may be significant comparing to the product tolerances and must be taken into account during simulation of such processes. Similarly the spring back of the cold forged part cannot be predicted and controlled without calculation of the elastic components of strain during the forming operation.

Many commercial FE codes include options for coupled thermal and mechanical simulation of metal forming processes. Such packages as ABAQUS, LS-DYNA, PAMSTAMP, LARSTRAN, FORGE and others during last years have realized different approaches to these problems.

To be able to deal with these tasks it is necessary to extend the model of the deformed material to visco-elasto-plastic behavior and to provide the material flow simulation coupled the tools deformation as it is described in present work.

Constitutive equations and models

Rigid viscoplastic formulation. If material is considered as incompressible rigid-plastic continua and elastic deformations are neglected the system of governing equations includes:

equilibrium equations:

$$\sigma_{ij,j} = 0 , \tag{1}$$

compatibility conditions:

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

constitutive equations:

$$\sigma'_{ij} = \frac{2}{3}\frac{\overline{\sigma}}{\dot{\overline{\varepsilon}}}\dot{\varepsilon}_{ij} , \tag{3}$$

incompressibility equation:

$$v_{i,i} = 0 , \tag{4}$$

and expression for flow stress:

$$\overline{\sigma} = \overline{\sigma}(\overline{\varepsilon}, \dot{\overline{\varepsilon}}, T) , \tag{5}$$

where σ_{ij} and $\dot{\varepsilon}_{ij}$ – components of stress and strain-rate tensors, v_i – velocity components, σ'_{ij} – deviatoric stress tensor, $\overline{\sigma}, \overline{\varepsilon}, \dot{\overline{\varepsilon}}$ – effective stress, strain and strain-rate, respectively, T – temperature.

In equations (1)–(5) summation convention is used. Comma denotes a derivative with respect to the axis following it. The indexes *i* and *j* for three-dimensional problems vary from 1 to 3 and repeated subscript means summation.

Explicit and implicit procedure. The material forming process is simulated incrementally using either explicit or implicit method of integration. In explicit method the velocity field $\mathbf{v}_k$ obtained for the actual configuration of the body $\mathbf{X}_k$ at the instant of time t_k is used to calculate its new configuration $\mathbf{X}_{k+1}$ at the next instant of time $t_{k+1} = t_k + \Delta t_k$ as follows

$$\mathbf{X}_{k+1} = \mathbf{X}_k + \mathbf{v}_k \Delta t_k . \tag{6}$$

In implicit method the velocity field $\overline{\mathbf{v}}_{k+1}$ is calculated at some instant of time t_{k+1} when the configuration of the body is not known yet and subject to change during the iteration procedure

$$\mathbf{X}_{k+1} = \mathbf{X}_k + \overline{\mathbf{v}}_{k+1} \Delta t_k , \tag{7}$$

where $\bar{\mathbf{v}}_{k+1}$ is the certain velocity at the time increment that can be taken as an average during the time increment

$$\bar{\mathbf{v}}_{k+1} = \frac{1}{2}\left(\mathbf{v}_k + \mathbf{v}_{k+1}\right). \tag{8}$$

The use of implicit method increases the stability of the solution (Osakada, 1982) but requires preliminary setting of the time increment Δt_k before solving of the plastic problem. In some circumstances it may cause the finite element mesh degeneration for the configuration $\mathbf{X}_{k+1}$, that is calculated using expression (7) or penetration of some nodes into the tools. Special precautions are to be done to avoid such problems.

Elastic plastic formulation. In case when elastic strain components are to be taken into account the model described by the system of equations (1)–(5) is to be modified and complemented by elastic governing equations

$$\sigma'_{ij} = 2G\,\varepsilon'^{e}_{ij}, \tag{9}$$

$$\sigma_o = K\varepsilon_V, \tag{10}$$

where G is the shearing elastic module, ε'^{e}_{ij} is the deviatoric elastic strain tensor, σ_o is the mean stress, K is volumetric elastic module, ε_V is the volumetric elastic strain. Total strain is the sum of elastic ε^e_{ij} and plastic ε^p_{ij} strain

$$\varepsilon_{ij} = \varepsilon^e_{ij} + \varepsilon^p_{ij}. \tag{11}$$

Differentiation of equations (9)–(11) with respect to time helps to represent all the expressions in terms of strain-rate that in turn can be expressed through the velocity as written in (2). The derivatives of the stress can be approximated as follows:

$$\frac{d\sigma}{dt} \approx \frac{\Delta\sigma}{\Delta t} = \frac{1}{\Delta t}\left(\sigma - \tilde{\sigma}\right), \tag{12}$$

where σ is the stress at current instant of time, $\tilde{\sigma}$ is the stress at previous instant of time and Δt is the time increment.

Finally the expressions for elastic-plastic can be written as follows:

$$\sigma'_{ij} = \frac{2}{3}\,\frac{\bar{\sigma}}{\dot{\bar{\varepsilon}} + \frac{1}{\Delta t}\,\tilde{\bar{\varepsilon}}^{e}}\left(\dot{\varepsilon}_{ij} + \frac{1}{\Delta t}\,\tilde{\varepsilon}'^{e}_{ij}\right), \tag{13}$$

$$\sigma_o = K\Delta t\,\dot{\varepsilon}_V + \tilde{\sigma}_o, \tag{14}$$

where the sign `~` is used to denote the value of the function in previous instant of time and implicit integration is supposed to be used.

Coupled deformation. The deformed shape of the tools is the result of application of the load from the deformed body to the tool surface. This deformed shape can be used for the simulation of the material flow at the next time increment.

Schematic diagram of this procedure is shown on **Fig. 1**. Let u_i are the present displacements at time t^e_i. Difference between elastic solution u^*_i for current load P_i and this value is

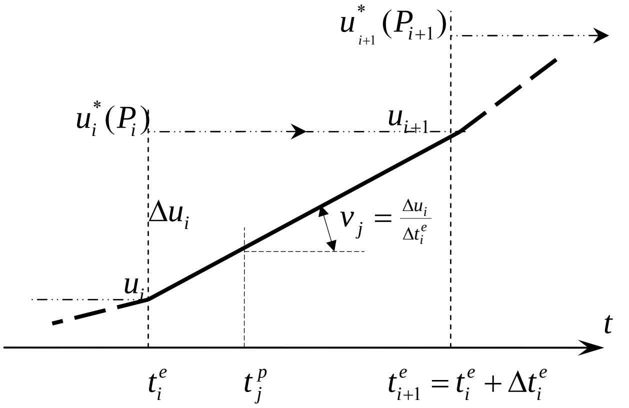


Figure 1. Schematic diagram for solution of couple elastic-plastic deformation problem.

$$\Delta u_i = u^*_i(P_i) - u_i. \tag{15}$$

Elastic solution u^*_i is to be found using expressions (9)–(10).

Additional velocities of tool points during time increment Δt^e_i could be defined as

$$v_j = \frac{\Delta u_i}{\Delta t^e_i}. \tag{16}$$

Value of Δt^e is independent with time increment for solution of plastic problem.

The velocity of any point on the tool surface is the sum of its velocity as a rigid body plus the velocity of its elastic deformation v_j in the same point. Thus synchronous displacement of workpiece and tool surfaces is guaranteed. Appropriate time step gives the value of additional velocity v_j that does not exceed the tool velocity as rigid body.

With such approach it is not necessary to combine all deformed bodies into a single system that may significantly increase the simulation time. Even though with this approach we get additional inaccuracy caused by the use of the load from previous configuration the error is supposed to be insignificant due to use of small time increments.

Coupled thermal problem. Energy balance equation for thermal problem in workpiece is

$$\rho c\dot{T} = (kT_{,i})_{,i} + \beta\bar{\sigma}\dot{\bar{\varepsilon}}, \tag{17}$$

where β – heat generation efficiency ($\beta = 0.9 \div 0.95$), ρ – density, c – specific heat and k – thermal conductivity.

Similar equation but without the last term is valid for the thermal problem in the tools where no heat is generation due to plastic work inside of the body.

Coupled simulation is performed by sequential solving of the thermal problem in the workpiece and then in the tools using actual boundary conditions on their surfaces that include surface heat flux due to convection and radiation and heating up due to friction.

Assembled dies, multiple deformed bodies. When simulating the tools set consisting of several parts assembled together (like pre-stressed dies, for example) or deformation of multiple plastic bodies consisting of different materials (bimetallic workpiece or several deformed bodies during rivet joining operation) an additional condition is to be applied on the adjacent surface of the contacting bodies. It is the non-penetration condition of the bodies inside each other that can be expressed as the following:

$$v_n^1 = v_n^2 , \qquad (18)$$

where v_n^i is normal component of velocity or displacement of the body number i. Numerical realization of this condition requires special finite element (**Fig. 2**), consisting of the nodes and the triangle that contacts this node. The normal load N that takes place on the contact is calculated using penalty method

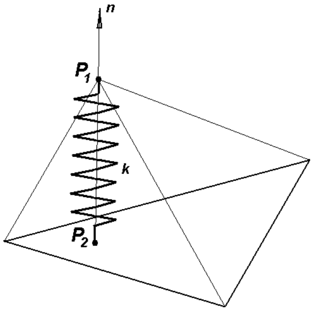


Figure 2. A special contact penalty element.

$$N = k(v_n^1 - v_n^2) , \qquad (19)$$

where k is positive number that is essentially bigger than diagonal terms of the stiffness matrix.

The contact conditions in tangent direction are realised through Coulomb friction law for the contacting parts of the assembled tools

$$T = \mu N , \qquad (20)$$

where T is nodal friction force, μ is friction coefficient, and constant friction law for the zones of plastic deformation

$$\tau = m\bar{\sigma} / \sqrt{3} , \qquad (21)$$

where τ is friction share stress at contact surface, m is friction factor.

Sample of simulation

The basic models of plastic, elastic and thermal behavior of the workpiece and tools described by equations (1)–(5), (9)–(10) and (17) have been verified several time for different conditions (Biba, 2008, Biba, 2010, etc.).

Turbine blade forging simulation using coupled deformation and coupled thermal problem. The turbine blade CrNiMoWV is produced in two forging blows as shown on **Fig. 3**. In the first blow the cylindrical billet has been upset and then it is forged in a screw press with flash


(a) (b)

Figure 3. The billet after the first (a) and second (b) forging blows.

The simulation has been performed using two approaches:

- Non-coupled model when the dies are considered as rigid bodies having certain uniform temperature,
- Mechanically and thermally coupled model where the die deformation and the temperature distribution in the tool is calculated concurrently with the deformation of the billet.

The simulation with both approaches shows that the die cavity surface deflection is non uniform with the maximum in the central part of the die (**Fig. 4**). Meanwhile in case of use the rigid die model such deflection does not influence the shape of the forged part that causes significant inaccuracy of the results. On the other hand when using the coupled model the forged blade gets the shape and dimensions that are superposition of the tool deformation due to contact normal pressure developed in forging process and thermal expansion caused by non-uniform temperature field. Resulting die deflection is shown on **Fig. 4c** where two crosscuts are shown along and across the blade. As we can see the total deflection of both dies in the centre is about 0.5 mm while the total blade thickness is 8 mm. Thus the total discrepancy with non-coupled simulation is more than 6%. On the other hand the deflection in the flash land area is two times smaller (0.24 mm).

It means that by adjustment of the press tooling set to the correct final distance it is impossible to compensate the die deflection at all parts of the die surface and the die cavity shape is to be profiled. It is also interesting that in case of coupled simulation the flash is smaller due to expanding of the die cavity.

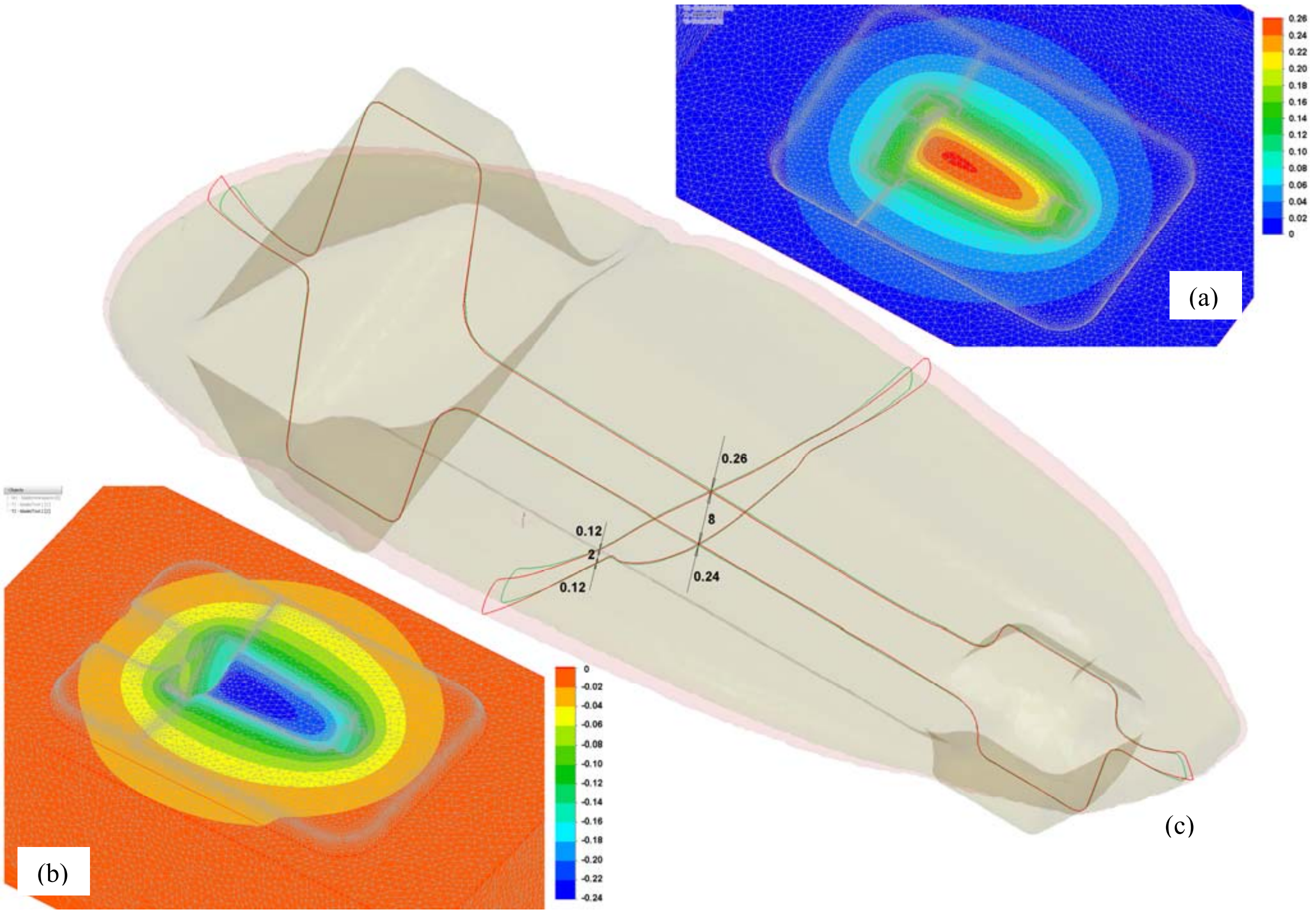


Figure 4. Displacement (in mm) in vertical direction of the upper (a) and lower (b) dies and the crosscuts of the blade with the contours for uncoupled and coupled simulation (c).

To estimate the accuracy of coupled procedure the tool calculation in postprocessor procedure when deformation load are applied to undeformable dies was done. Obtained difference with coupled solution at any time is a little bit smaller postprocessor results.

The temperature in the lower die obtained by coupled simulation is shown on **Fig. 5**. Similar distribution of the temperature is in the upper die.

Figure 5. The temperature in the lower die at the end of the forging blow.

Temperature distribution has significant variation and this information could be very useful for evaluation of the tool life span that is influenced by abrasive die wear, thermal cracks and stress concentration in critical areas.

Assembled dies analysis when forging an aluminium fork. The forging of the aluminium fork (**Fig. 6**) made of AA6101 is performed using cylindrical billet in one blow.

The upper die (**Fig. 7**) consists of two assembled parts (1, 2) that are put in contact before the operation and are kept in this position by specified load during forming operation. Each part is also assembled and includes the die holder (1a, 2a), die insert (1b, 2b) and ejector (1c). The billet 3 is deformed by the punch 4 going upwards.

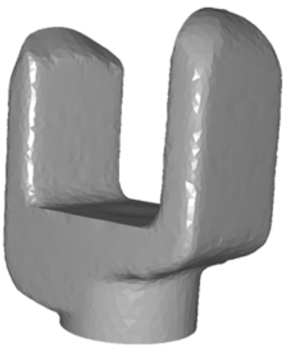
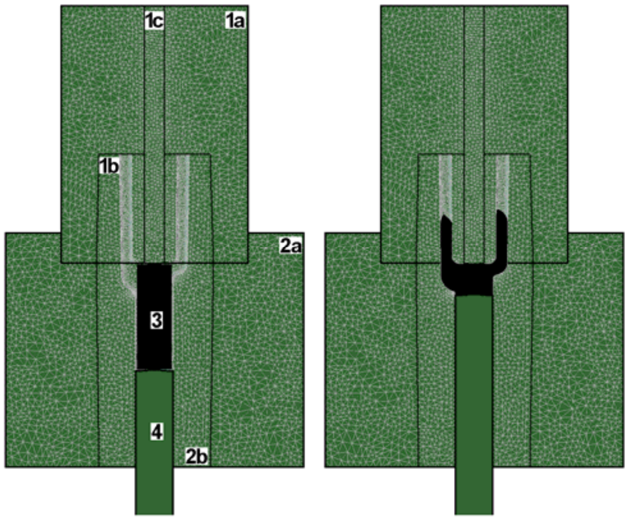


Figure 6. The forged part.



(a) (b)

Figure 7. The tooling set at initial (a) and final (b) position.

Due to the symmetry of process only a half of workpiece has been taken for simulation.

The results of the simulation of the assembled tooling set are shown on **Fig. 8**. Its parts can slide over each other along the contact surfaces and may separate in case if the holding load is not sufficient.

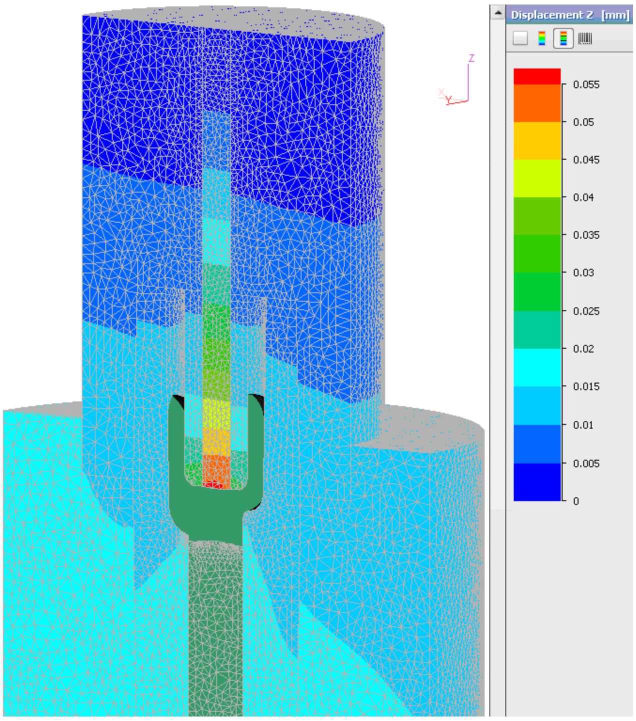


Figure 8. Distribution of vertical displacement in the assembled tooling set.

Fig. 9 shows the distribution of the effective stress in the tooling set where the most stressed areas can be observed in the die cavity. Applying compression force by using shrink fitting it is possible to reduce tensile stress in these areas and to reduce probability of the cracks due to fatigue.

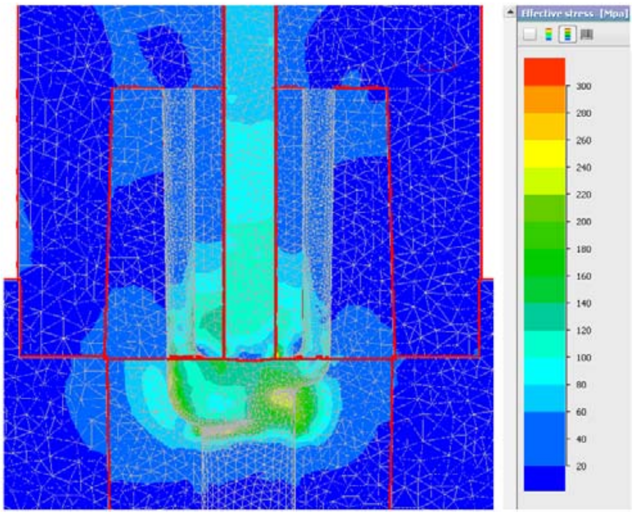


Figure 9. Distribution of effective stress in the central part of the assembled tooling set (the billet and punch are removed).

Conclusion

1. The mechanically and thermally coupled model has been developed and tested in a new version 7 of QForm3D software.
2. The tests have shown that elastic deformation of the dies may have great influence on the accuracy of the simulation when forging the parts with high requirements to finish geometry like turbine blades.
3. Thermal coupling allows more accurate calculating of the distribution of the temperature in the dies and workpiece and provides the background for prediction of the thermal cracks and fatigue in case if respective material models can be incorporated.

References

Biba N., Stebunov S., Lishny A., Vlasov A. New approach to 3D finite-element simulation of material flow and its application to bulk metal forming, 7th International Conference on Technology of Plasticity, 27 October – 1 November, 2002, Yokohama, p. 829–834.

Osakada, K., Nakano, J., Mori, K. Finite element method for rigid-plastic analysis of metal forming – formulation for finite deformation, Int. J. Mech. Sci., 1982, N 8, p. 459–468.

Biba N., Stebunov S., Vlasov A. Material forming simulation environment based on QForm 3D software system. 12th International Conference Metal Forming 2008, 21–24 September, 2008. Vol. 2, p. 611–616.

Biba N., Stebunov S. QForm3D – cost effective simulation tool for metal forming technology. Proceedings of the Korean Society for Technology of Plasticity Conference, 15th Forging Symposium, 10–11 June, 2010, Changwon, South Korea, p. 77–80.