The model for simulation of thermally, mechanically and physically coupled problems of metal forming

Nikolay Biba, Alexander Maximov, Sergei Stebunov, Alexey Vlasov

QuantorForm Ltd., Moscow, Russia, info@qform3d.com

Abstract. The paper presents the latest 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 and equipment components as well as with the evolution of material properties. The dies in coupled simulation 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 performance of the model is illustrated by practical example using an industrial case study.

Keywords: Forging, Simulation, Coupled problems, Press, Finite Element

1. GENERAL

The simulation of hot metal forming processes in most cases can be done using an assumption that the deformed material is a rigid-plastic continuum while the dies are rigid bodies as presented in our work [1]. 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.

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 we explained in [2]. In present work the limits of coupling has been extended towards including into consideration not only the tools deformation but also the components of forging equipment, several deformable bodies and internal parameters of the material.

2. CONSTITUTIVE EQUATIONS AND MODELS

2.1. Rigid and elastic visco-plastic formulation for deformable workpiece

Deformable material is considered as incompressible rigid-plastic continuum where elastic deformations could be neglected. Constitutive equations for this material are based on Levi – von Mises law

$$\sigma'_{ij} = \frac{2}{3}\frac{\bar{\sigma}}{\dot{\bar{\varepsilon}}}\dot{\varepsilon}_{ij} \tag{1}$$

where σ_{ij} and $\dot{\varepsilon}_{ij}$ are components of stress and plastic strain-rate tensors, σ'_{ij} is deviatoric stress tensor, $\bar{\sigma}, \dot{\bar{\varepsilon}}$ are effective stress and strain-rate in deforming body respectively.

Expression for flow stress σ is presented as

$$\sigma = \bar{\sigma}(\bar{\varepsilon}, \dot{\bar{\varepsilon}}, T) \tag{2}$$

where $\bar{\varepsilon}$ – effective strain, T – temperature.

In case when elastic strain components are to be taken into account the model is to be modified and complemented by elastic governing equations (Hook's law)

$$\sigma'_{ij} = 2G\varepsilon'_{ij} \quad \text{and} \quad \sigma_0 = K\varepsilon_V \tag{3}$$

where G is the shearing elastic module, ε'_{ij} is the deviatoric elastic strain tensor, σ_0 is the mean stress, K is volumetric elastic module, ε_V is the volumetric elastic strain.

2.2. Coupled deformation

One of the problems for solution of coupled elastic and plastic deformations consists in different terms that are used for description of behavior. Plastic behavior is formulated in terms of stress-strain rate (1) and elastic behavior is formulated in terms of stress-strain (3). Another problem is significant calculation time required in the case of coupling of all plastic and elastic deformed objects into a single system. To overcome this problem another approach was suggested.

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 **Figure 1**. Let u_i be the present displacements at time t_i^e. Difference between elastic solution u_i^* for current load P_i and this value is

$$\Delta u_i = u_i^*(P_i) - u_i \tag{4}$$

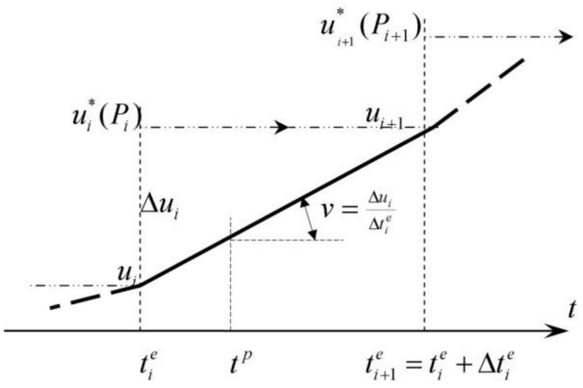


Figure 1. Schematic diagram for solution of coupled elastic-plastic deformation problem.

Elastic solution u_i^* is to be found using expressions (3). Additional velocities of tool points during time increment Δt_i^e could be defined as

$$v_{\text{add}} = \frac{\Delta u_i}{\Delta t_i^e}, \tag{5}$$

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

The velocity v_{sum} of any point on the tool surface is the sum of its velocity as a rigid body v_{die} and the velocity of its elastic deformation v_{add} in the same point:

$$v_{\text{sum}} = v_{\text{add}} + v_{\text{die}}, \tag{6}$$

Thus synchronous displacement of workpiece and tool surfaces is guaranteed. Appropriate time step gives the value of additional velocity v_{add} that does not exceed the tool velocity as rigid body.

With such approach it is not necessary to merge all deformed bodies into a single system that may significantly increase the simulation time especially for plastic iteration procedure. 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

$$\rho c\dot{T} = (kT_{,i})_{,i} + \beta\bar{\sigma}\dot{\bar{\varepsilon}}, \tag{7}$$

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

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

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 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:

$$u_n^1 = u_n^2, \tag{8}$$

where u_n^i is normal component of displacement of the body number i.

Numerical realization of this condition requires special finite elements (**Figure 2**), consisting of the contacted node $\mathbf{P^1}$ and the basis (linear or triangle) with this node projection point $\mathbf{P^2}$. The normal load P_n that takes place on the contact is calculated using penalty method

$$P_n = c(u_n^1 - u_n^2), \tag{9}$$

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

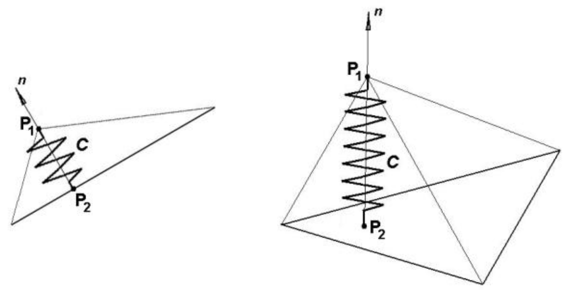


Figure 2. Special contact penalty elements for 2D (left) and 3D (right) problems.

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

$$P_t = \mu P_n, \tag{10}$$

where P_t is nodal friction force, μ is friction coefficient.

2.3. Physical coupling and User's Defined Functions (UDF)

User's defined functions are intended to calculate the parameters and functions that are not initially included in the list of standard functions. Some of them can be dependent on other UDF and/or can be used for calculation of some parameters that may influence the material behavior. For example, by means of UDF it is possible to calculate grain size evolution due to dynamic and static recristallisation that in turn can be used as a parameter for calculation of the flow stress. Particularly in this work the flow stress though not dependent on grain size has been programmed as UDF using a script to substitute the following expression:

$$\sigma = 1150\,\bar{\varepsilon}^{0.07}\exp(-0.07\bar{\varepsilon})\,\dot{\bar{\varepsilon}}^{0.15}\exp(-0.0027T) \quad (11)$$

The script has been written in Lua that is a powerful, fast and lightweight embeddable scripting language [3]. QForm 7 may embed and execute Lua scripts to calculate any UDF and on the contrary to FORTRAN subroutines no compiler is needed. The script to calculate the flow stress data according to expression (11) is presented in **Table 1** where A, m1,..m4 are material parameters and correspondingly their numerical values.

Table 1. Lua script to define the flow stress expression.

```
A  = parameter("A",  1150)
m1 = parameter("m1", 0.0027)
m2 = parameter("m2", 0.07)
m3 = parameter("m3", 0.15)
m4 = parameter("m4", 0.07)
pow = math.pow
exp = math.exp
function FlowStress(T, strain, strainRate)
s = strain
r = strainRate
F = A * exp(-m1*T) * pow(s,m2) * pow(r,m3) * exp(-m4*s)
return F
end
```

3. RESULTS AND DISCUSSION

The basic models of plastic, elastic and thermal behavior of the workpiece and tools described above have been verified for different conditions and published, for example, in [4,5]. Here we present the case how the deformation of the forged part can be coupled with the deformation and temperature in the dies and in the forging press components.

As an example we considered a new 10 MN universal hydraulic press installed in TU Bergakademie Freiberg, Germany [6]. The geometric model of the press frame and other components subject to forging load has been produced in SolidWorks. The model saved in universal STEP format has been imported to QForm 7 and then FE mesh has been generated for all the components to be modeled. Totally the press structure having quite detailed shape of the components required the mesh with about 90’000 of nodes and 390’00 of tetrahedral elements while the number of the nodes in a forged part varies from about 4000 nodes at the beginning of deformation to 30’000 nodes at the end (**Figure 3, a**). After conversion of geometric model to FE representation the other parameters of the equipment and materials have been specified. They include kinematics of the machine, mechanical and thermal properties of forged material and material of machine components.

The simulation has been performed for the sample part made of 16CrMn4 steel. Initial temperature for the billet was 1150°C, for the die block 250°C, for the frame and the ram 20°C. Feet of press frame were fixed. Press was loaded by the deformation force and pressure in hydraulic cylinder. Properties of all equipment parts were the same having Young’s modulus $2.16\cdot10^5$ MPa and Poisson’s coefficient 0.33.

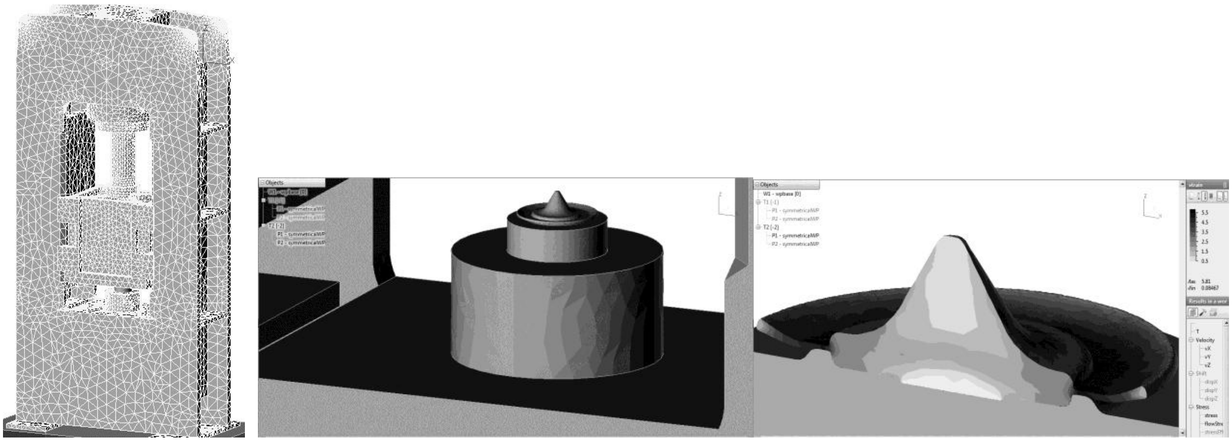
Forging simulation has been performed till specified final thickness of the flash that is 3.0 mm that provides complete filling in the die as shown on **Figure 3,b**. Strain distribution in the forged part at the end of the forging stroke is shown in **Figure 3, c**.

The results of coupled thermo-mechanical simulation in terms of displacement are shown in **Figure 4a**. As clearly seen all parts of the press structure are subject to elastic deformation. The biggest absolute displacement can be observed in the upper die that actually also accumulates the displacement of the ram and piston shaft as well. The displacement (deflection) of the die itself is small and does not exceed 0.1-0.15 mm. Meanwhile the deformation of the ram (**Figure 4, b**) and the piston shaft totally may reach 0.8 mm. This displacement value is to be taken into consideration when setting up the press for forging. Dividing total forging load that was calculated by means of simulation as about 8 MN by the total displacement value we can get overall stiffness of the press that in our case can be estimated approximately as 10MN/mm. This value of stiffness can be used as a useful parameter to estimate the influence of the forging load variation on the accuracy of the forged parts.

Figure 4 c shows the temperature distribution in the forged part, the dies, the die holders and the ram. This simulation has been done for nearly steady state of the tooling set that can be reached after several forging cycles. As seen from the picture the thermal field in the die and the die holder is not uniform that in turn causes additional deflection of the tooling set that also is to be taken into account to ensure the best conditions for precise forging technology.

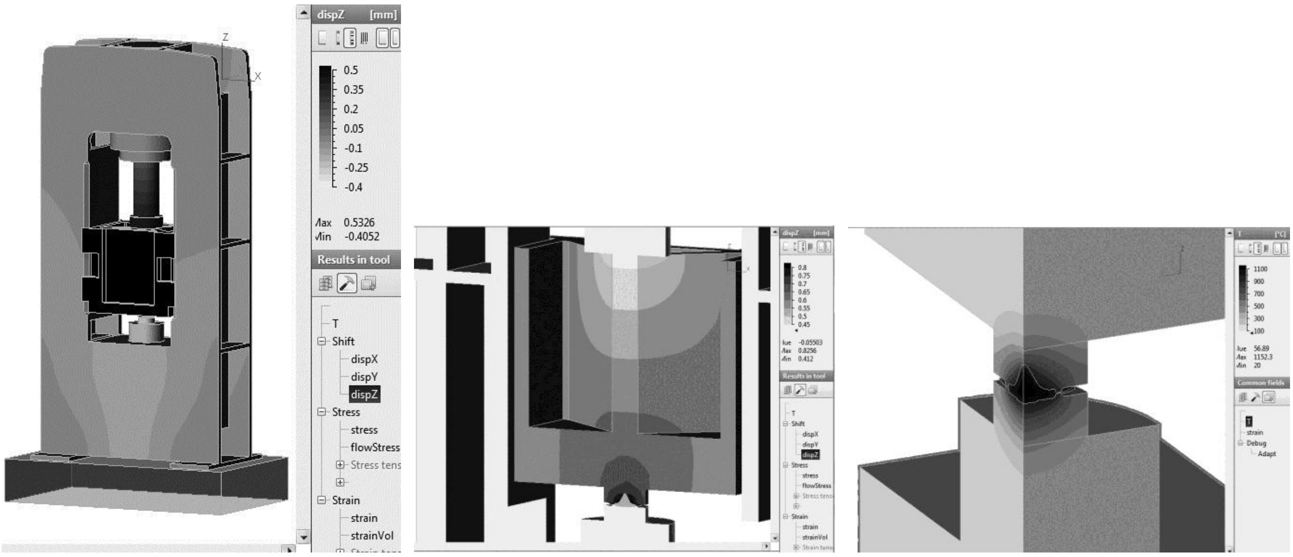
Further coupled analysis of the system of the press and the forged billet may be extended towards including into consideration the dynamics of hydraulic drive depending on the forging load.

.



a) b) c)

Figure 3. General view of the press structure as a FE model starting position of deformation (a), forged part on the lower die after forging blow (b), the strain distribution in the meridian cross cut of the forged part (c).



a) b) c)

Figure 4. Vertical displacement distribution in mm shown over the press structure (a) and in the crosscut of the upper die and the ram (b); temperature distribution in crosscut of the forged part, dies and die holders

4. CONCLUSIONS

1. The mechanically and thermally coupled model of the press and the forged billet has been developed and tested in a new version 7 of QForm3D software.
2. The tests have shown that elastic deformation of the dies is relatively smaller than overall deflection of the equipment components that may have great influence on the accuracy of the forging.
3. Additional coupling of a thermal problem allows more accurate calculation of the heat exchange between the billet and the tooling set and in the same time provides data for accurate calculating of the deflection of die and the ram that is important for precision forging.

5. REFERENCES

[1] N. Biba, S. Stebunov, A. Lishny, A. Vlasov: New approach to 3D finite-element simulation of material flow and its application to bulk metal forming, 7th International Conference on Technology of Plasticity, Yokohama, ed. M. Kiuchi (2002), 829–834.

[2] S. Stebunov, N. Biba, A. Vlasov, A. Maximov: Thermally and Mechanically Coupled Simulation of Metal Forming Processes in Proceedings of the 10th International Conference on Technology of Plasticity, Aachen, eds. G. Hirt, E. Tekkaya (2011), 171-175.

[3] R. Ierusalimschy: Programming in Lua, Publishing House of Electronic Industry, Beijing (2008).

[4] N. Biba, S. Stebunov, A. Vlasov: Material forming simulation environment based on QForm 3D software system. 12th International Conference Metal Forming, 2 (2008), 611–616.

[5] N. Biba, S.Stebunov: QForm3D – cost effective simulation tool for metal forming technology. Proceedings of the Korean Society for Technology of Plasticity Conference, 15th Forging Symposium, Changwon (2010), 77–80.

[6] A.Vlasov, G. Lehmann, S. Dedov: Coupled Simulation of the System "Workpiece – Dies – Forging Press" by Finite Element Model, MEFORM 2012, Freiberg, Germany (in publication).