The Simulation of the Extrusion Process Using QForm3D

N. Biba*[1], A.Vlasov[2], S. Stebounov[1]
[1] QuantorForm Ltd. P.O. Box 39, 119049, Moscow, Russia, e-mail: info@qform3d.com
[2] Moscow State Industrial University, Avtozavodskaya st. 16, 115280 Moscow, Russia, e-mail: tlanda@td.lpi.ac.ru
* Corresponding Author

INTRODUCTION

The die design for extrusion of the profiles having complicated cross sectional shapes is still an art rather than a science. Die design for a new product is usually being developed on the basis of previous experience and experimentation. Meanwhile in many cases costly experiments and in-plant trials can be replaced by numerical simulation. It is effective for analysis of the material flow, temperature distribution and load estimation. From practical point of view the most important are the stages of the die filling when the front end of the profile is formed and the steady-state stage when the pulling force is applied and the product shape and properties are formed. At the first stage the welding zones are formed and their location and propagation are very important for the product quality. To analyse the welding conditions the stress and deformations in welding areas are to be controlled. During the steady-state stage of the process some parameters (the punch force, the temperature) may vary but this variation does not influence the material flow considerably.

The paper presents 3D finite element system for simulation of extrusion of the profiles including the profiles of hollow shapes. For the first stage the non-steady-state simulation is performed while the second stage is simulated by non-isothermal steady-state flow approach using the visco-plastic material in the container and in the die outlet (Euler approach). The developed code provides completely automated mesh generation, effective simulation of the material flow and output of the information on material flow, temperature distribution and some integral characteristics like profile bending and profile twisting which are necessary for prediction of the product quality. The use of tetrahedral elements provides simulation of the most complicated profile shapes keeping the total amount of the elements within several tens of thousands. The geometrical source data are imported as a solid model from CAD systems. The developed software was applied to simulation and optimization of different extrusion processes including profiles having solid and hollow shapes. The contact stresses on the die-billet interface are used for the simulation of the tooling set including its stress state and the deflection. The code is the effective tool for extrusion technology development in industry due to its user-friendliness. It was validated by industrial use and has allowed improving the extrusion technology for many practical cases.

MATERIAL FLOW FORMULATION

The numerical model for FEM simulation is based on flow formulation [1] where the material is considered as incompressible rigid-viscoplastic continua and elastic deformations are neglected. The system of governing equations includes:

dynamic equations

$$\sigma_{ij,j} = 0 \qquad (1)$$

compatibility conditions

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

constitutive equations

$$\sigma_{ij} = \frac{2}{3}\frac{\bar{\sigma}}{\dot{\bar{\varepsilon}}}\dot{\varepsilon}_{ij} \qquad (3)$$

incompressibility equation

$$v_{i,i} = 0 \qquad (4)$$

energy balance equation

$$\rho c\dot{T} = (k\,T_i)_{,i} + \beta\bar{\sigma}\dot{\bar{\varepsilon}} \qquad (5)$$

and flow stress given by equation

$$\bar{\sigma} = \bar{\sigma}(\bar{\varepsilon}, \dot{\bar{\varepsilon}}, T) \qquad (6)$$

where σ_{ij}, $\dot{\varepsilon}_{ij}$, v_i are the stress, strain-rate and velocity components respectively, s_{ij} is the deviatoric stress tensor, $\bar{\sigma}, \bar{\varepsilon}, \dot{\bar{\varepsilon}}$ are the effective stress, effective strain and effective strain-rate, respectively, T is the temperature, β is the heat generation efficiency which is usually assumed as $\beta = 0.9 - 0.95$, ρ is the density, c is the specific heat and k is the thermal conductivity.

Friction model proposed by Levanov et al [2] is used on the contact part of workpiece surface

$$F_t = m\frac{\bar{\sigma}}{\sqrt{3}}\left(1 - \exp(-1.25\frac{\sigma_n}{\bar{\sigma}})\right) \qquad (7)$$

where m is the friction factor, σ_n is the normal contact pressure. Expression (7) can be considered as a combination of constant friction model and Coulomb friction model that inherits advantages of both ones. The second term in parenthesis takes into account the influence of normal contact pressure. For high value of contact pressure expression (7) provides approximately the same level of friction traction as constant friction model while for low contact pressure it gives friction traction that is approximately linearly dependent on normal contact stress. Experimentally determined values of m for many lubricants, materials and surface conditions in hot and cold state were experimentally obtained by Levanov and can be found in [2].

Equations (1-4) were transformed into discrete form by means of virtual work-rate principle and finite element technique. Velocity and mean stress fields are approximated by tetrahedral elements for 3D [3].

RESULTS AND DISCUSSION

3.1 Steady-state simulation

Finite element mesh is generated automatically for space domain that includes the billet in the container and the part of the product having sufficient length to be rigid at its front end. Product shape is arbitrary. Source data for mesh generation includes the following geometry models: the container, the die, the mandrel, the billet and the punch. The die may be combined with the container and the mandrel in a single solid body.

The geometric model from CAD system is transferred to the program through STEP or IGES file format as a solid body or surface model. The type of the model depends on the CAD system. The geometry transfer through STEP or IGES provides its conversion to FE representation without losing the accuracy and smoothness of the model. The code for 3D-extrusion model is realised as an extension of commercial software system QForm3D. The program has the reliable interface to CAD systems, provides fully automatic 3D FE mesh generation in the dies and the billet and completely automatic simulation of the profile extrusion process. Limited amount of source data provides fast and easy work even for inexperienced users.

After solving the visco-plastic problem using the steady-state approach we have the distribution of the different fields inside of the deformed body and the components of the velocity vector in the nodes of the FE mesh. Using the values of the velocity vector inside of the profile it is possible to calculate the profile bending in mm per one meter of the profile length; the angle of the profile twisting per one meter of the profile length; possible deflection of the profile shape.

The algorithm of calculation of these characteristics consists of two stages: prediction of the shape of the profile just after its exit from bearing zone and calculation of the integral characteristics of shape like the bending and twisting using the FE mesh in the nodes. For the end of the profile just in the exit from bearing zone the minimum velocity $V_{z\,min}$. and the maximum velocity V_{max} are calculated (Figure 1). Then for every node we can calculate the time increment for the movement of the material particle along Z direction $dt=dz/V_{max}$. Then using this time increment dt it is possible to calculate the displacement of the each point of the profile after leaving the bearing zone in the X and Y directions using the following impressions:

$$dX_{i+1}=dX_i+dt\ V_{x\ i+1} \qquad (8)$$

$$dY_{i+1}=dY_i+dt\ V_{y\ i+1} \qquad (9)$$

Where i – is the number of the node along flow line.

Because the velocity component V_z is changing in cross section of the profile it is necessary to apply the averaging procedure for calculation of the displacement along Z direction:

$$dZ_{i+1}=dZ_i+dt\ (V_{z\ i+1}-V_{z\ min}) \qquad (10)$$

It is evident that the described algorithm is not absolutely accurate. But it satisfies two important requirements:

the displacement dX_{i+1}, dY_{i+1} and dZ_{i+1} in case of no bending and twisting of the profile will be equal to zero;

the value of the displacement of the nodes in X, Y and Z directions is linear with respect to the velocity in these directions.

Let us consider the die design optimization for extrusion the profile from aluminum alloy AISI AA6101 for building industry. Its drawing is shown of Figure 2. Extrusion is performed in flat die with feeder plate. The purpose of optimization with help of simulation was to find out the shape of the feeder plate and the profile position that provides the minimum twisting. On the Figure 3 are shown three variants of the shapes of the feeder plate and positions of the profile on the die. The depth of the feeder plate is 15 mm. The length of the billet at this stage is 100 mm. The speed of the punch is 4 mm/s. The elongation ratio for this profile is 130. Temperature of the billet is 460 °C.

After calculation the twisting and bending for all variants the best is the variant shown on Figure 3c (4.9 radian/m). The experiment value of the twisting for this die was 3.2 radian/m (see Figure 3d).

The approach described in this topic seems effective because of combination of full automatic steady-state profile extrusion simulation with reliable calculation of twisting and bending of the profile. Further development of this approach lies in the way of experimental verification and accumulation statistics during its industrial testing.

3.2 Non steady-state simulation

The following simulation was performed using a non-steady-state mode. It is the extrusion of hollow aluminium profile (tube) thus the metal flow separates and then merges in welding zones. Initial temperature of the billet was 490°C, the die and the container temperature 460°C. Finite element mesh was generated automatically for the dies and the billet using their original shape as a source data. The FE mesh is generated in QForm in the tool and in the billet fully automatically without user's interference.

The main goal of simulation of initial stage of deformation was to analyze the material flow when it fills the channels in the tooling set near the mandrel and to find the zones where separated streams of the material flow merge again creating welding areas. Figure 4 shows several sequential stages of the material flow. The areas where the surfaces of merging streams come in contact with each other are marked by the clouds of the spots. In these areas welding process takes place. It is seen that the welding zones are continually extended along the extruded product and due to high elongation they create the strips along the profile.

As soon as the profile leaves the die orifice it may bend and twist due to non uniform velocity of flow of different parts the profile that in turn depends on many parameters. The simulation shows the shape of the profile after it leaves the die as on Figure 5. The simulation shows increasing of the temperature of the extruded profile up to 506°C. It reaches the maximum in the die orifice and then rapidly drops down as soon as the material leaves the die.

Figure 6 shows the load on the punch versus distance between the punch and the mandrel. The graph has the increasing part at the stage when the material fills the cavities in the tooling set. Then the graph reaches the maximum value that corresponds to the beginning of the steady state stage of the deformation. This stage of the deformation lasts nearly stable till the end of the extrusion process when the whole billet passes through the die and the punch stops. The simulation time for the whole process using non-steady state model is unacceptably large but it is not necessary to make it because the maximum load value is reached at the beginning of the steady state stage.

The stress and the deformation in the combined tool that includes the container, mandrel and die cap were calculated for the position of the punch that corresponds to the maximum load value. The axial displacement reaches the maximum value 0.3 mm at the end part of the mandrel (Figure 7). Such deformation means that the bearing on the die and the mandrel will be shifted with respect to each other that may cause different material flow pattern comparing to presumption that the tooling set is a rigid body. In real practice such deformation must be compensated by the mandrel design and the data for such compensation can be obtained from simulation.

The effective strain distribution (see Figure 8) shows that its maximum value is near the root of the edges of the mandrel. Thus this zone must be considered as the most dangerous from the point of view of low cycle fatigue failure.

CONCLUSION

Both approaches described in the paper are effective for different kind of the analysis. While steady-state approach allows analysing the developed stage of the extrusion process, the non-steady-state method allows carrying out the comprehensive analysis of the initial stage of the process. Further development of both methods lies in the way of experimental verification and accumulation statistics during its industrial testing.

REFERENCES

[1] Zienkiewicz, O.C. “Flow formulation for numerical solution of metal forming processes”. *Numerical analyses of forming processes,* 1-44 1984. (*Ed. J.F.T. Pittman, O.C.Zienkiewicz, R.D.Wood and J.M.Alexander*):.

[2] A.N. Levanov, V.L. Kolmogorov, S.P. Burkin et al, Contact Friction in Metal Forming, Moscow, 1976 (In Russian).

[3] Lishny A.I., Biba N.V., Milenin A.A. “Two level approach to the problem of steady state extrusion process optimisation”. *Proceeding of Simulation of Materials Processing: Theory, Methods and applications, NUMIFORM’98*, 627-631, Enschede, Netherlands, 22-25 June 1998, A.A. Balkema, Rotterdam, Brookfield

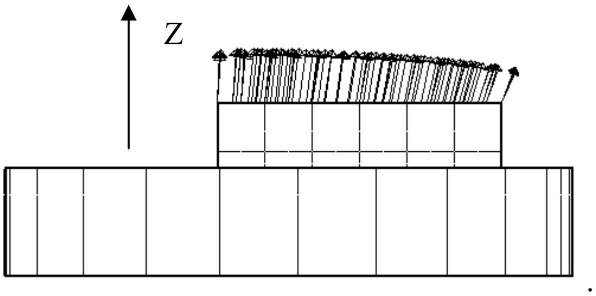


Figure 1: Scheme of the velocity vectors at the end of the product calculated by 3D-model.

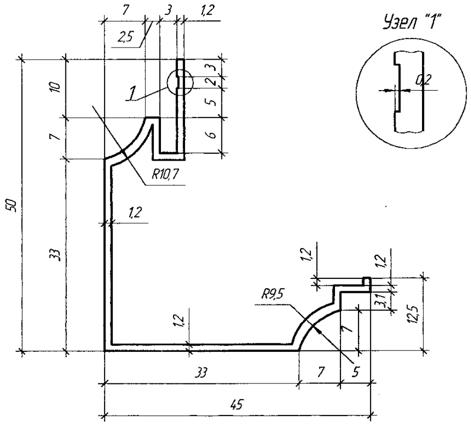
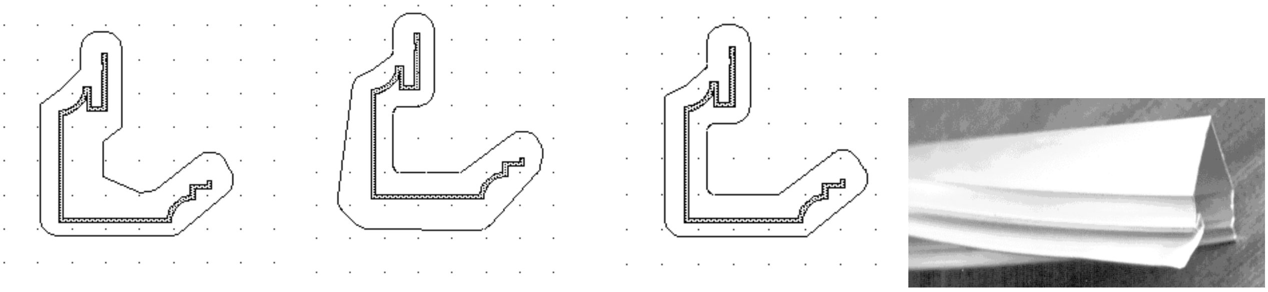
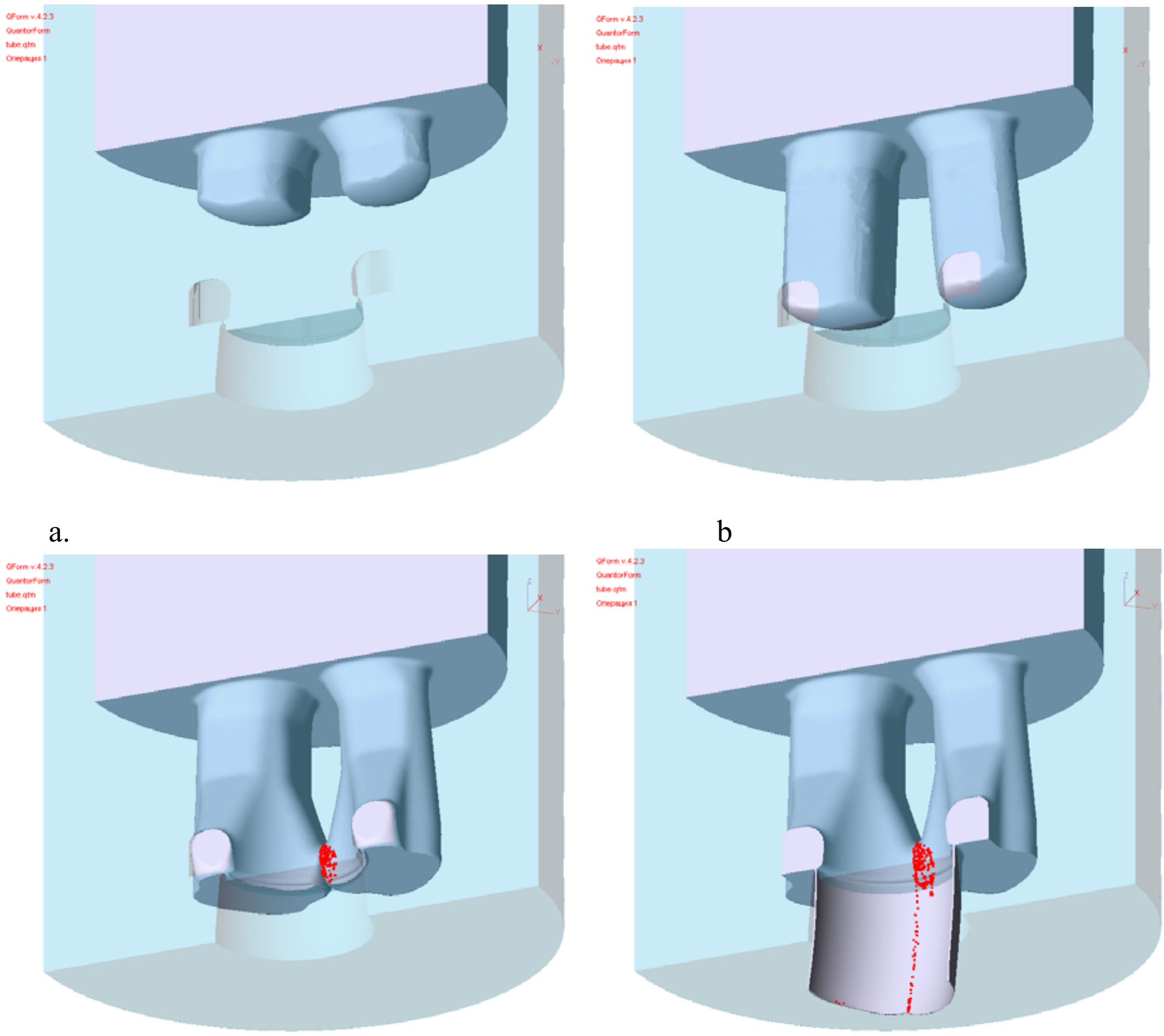


Figure 2: Drawing of the profile for building industry



a. b c d

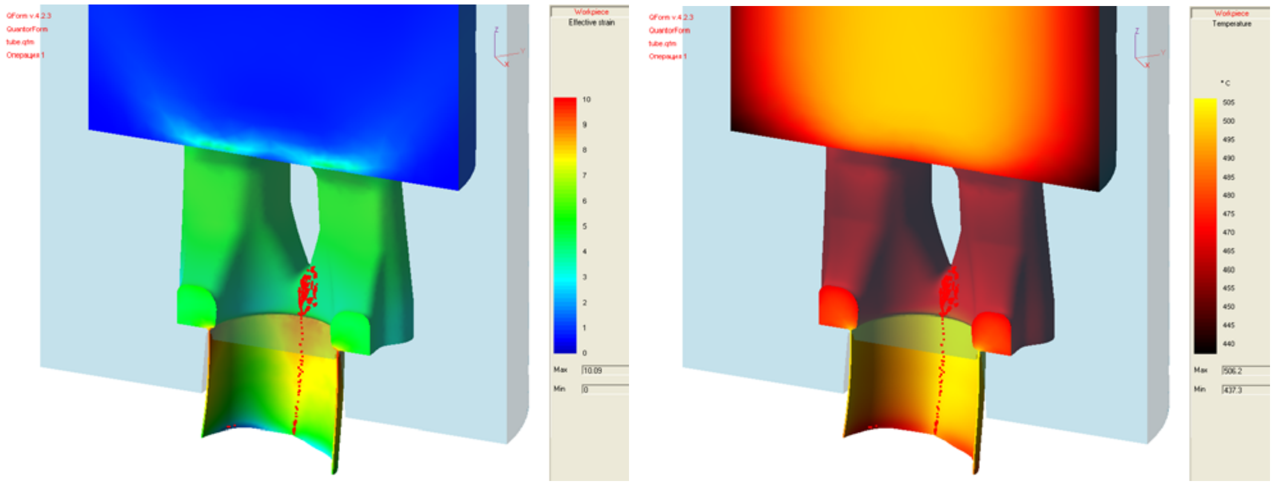
Figure 3: Three different variants of the of the feeder plate shape. Predicted profile twisting: a –7.6 radian/m, b – 5.75 radian/m and c – 4.9 radian/m, d – the experiment.



a. b

c. d.

Figure 4: Simulation of the initial stage of the tube extrusion: (a,b) –filling of the die, (c,d) – forming of the welding zone and the profile.



a. b.

Figure 5: The effective strain (a) and the temperature distribution (b).

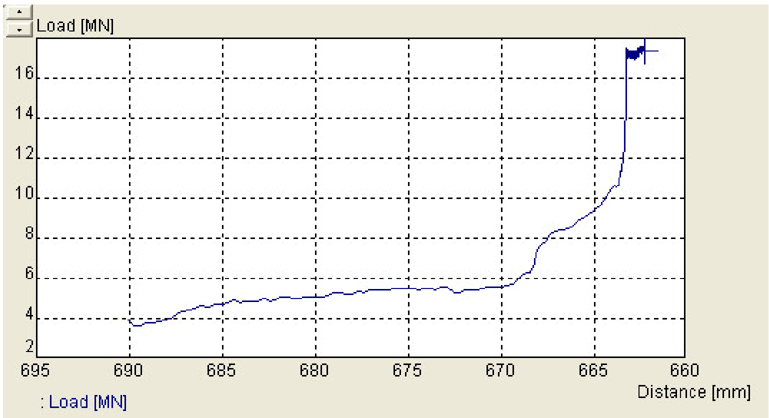


Figure 6: The graph of the load versus punch travel.

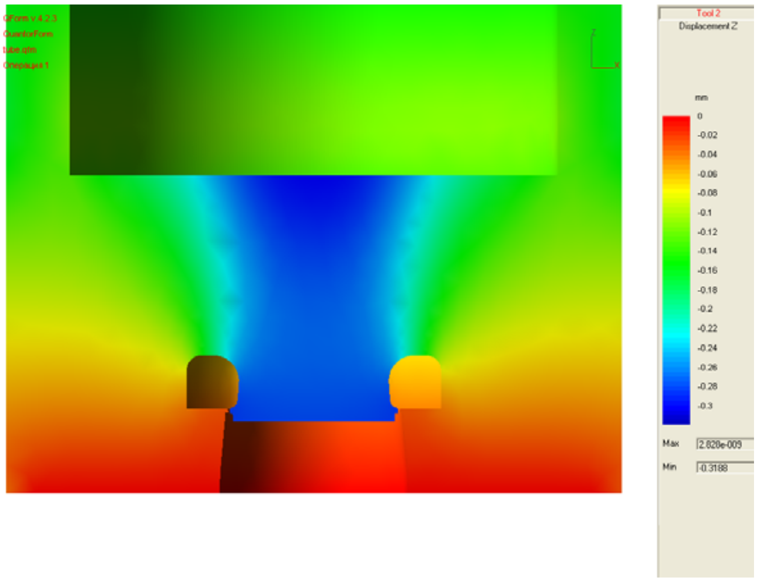


Figure 7: Deformation of the mandrel and the die in axial direction. The displacement in Z-direction is shown. Maximum displacement is 0.3 mm

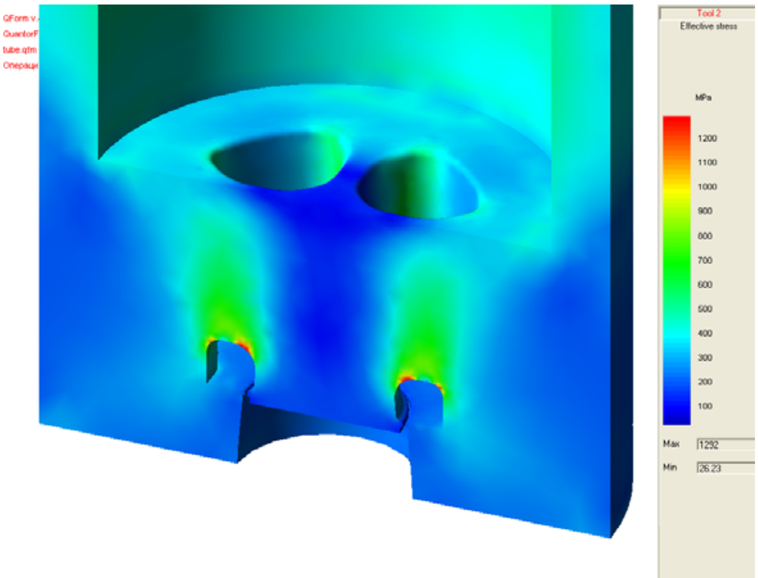


Figure 8: The effective stress distribution in the tooling set.