Journal for Technology of Plasticity, Vol. 28 (2003), Number 1-2

# FINITE ELEMENT MODELING OF HYDROSTATIC EXTRUSION OF MAGNESIUM

*R. Kopp, G. Barton,*
*Institute of Metal Forming - RWTH Aachen, Germany*

## ABSTRACT

*Hydrostatic extrusion is a metal forming process characterised by high extrusion ratios even with difficult to form materials. A hydrostatic compression stress state is characteristic for this process. It improves the formability and allows combined with the good lubrication homogeneous metal forming properties across the extruded cross sections at high forming speeds.*

*A simulation model of hydrostatic extrusion was developed and implemented into the FE package PEP & LARSTRAN/SHAPE. To deal with the large extrusion ratio of this process a templatere meshing technique using structured meshes has been integrated.*

*With the developed simulation model the material flow properties like strain rate, equivalent strain and temperature distribution are computed in a thermo-mechanical coupled simulation. The simulation model has been validated by comparison of the computed process parameters to parameters measured during experiments on an industrial hydrostatic extrusion press.*

## 1. INTRODUCTION TO HYDROSTATIC EXTRUSION

Figure 1 shows the principle of hydrostatic extrusion. Its main components are the die, the pressure medium and the billet, which will become the extruded product later during the process. The workpiece (billet) is placed inside a container, and has to be tapered to match the die geometry. A guiding plate is sometimes used to prevent the billet from tilting. The gap between the billet and the container is filled with a pressure medium. The pressure medium surrounds the billet and transforms the forming force on the billet. When the pressure of the pressure medium becomes high enough, the material is pressed through the die.

During extrusion, the pressure medium is forced by its inherent pressure into the gap between the die and the workpiece. There, it acts as a lubricant effectively reducing the friction forces. The pressure medium is drawn by adhesion to the surface of the workpiece out of the die and can be found on the extruded profile. The low friction force caused by this lubrication and the conical die geometry causes high hydrostatic compression stress in the deformation zone of the material, which is favourable for many difficult to extrude materials.

There are several differences between the hydrostatic extrusion process and (conventional) direct forward extrusion process, which are relevant for the modeling:

- the forming force is applied by a pressurised medium and has to be represented in the simulation model

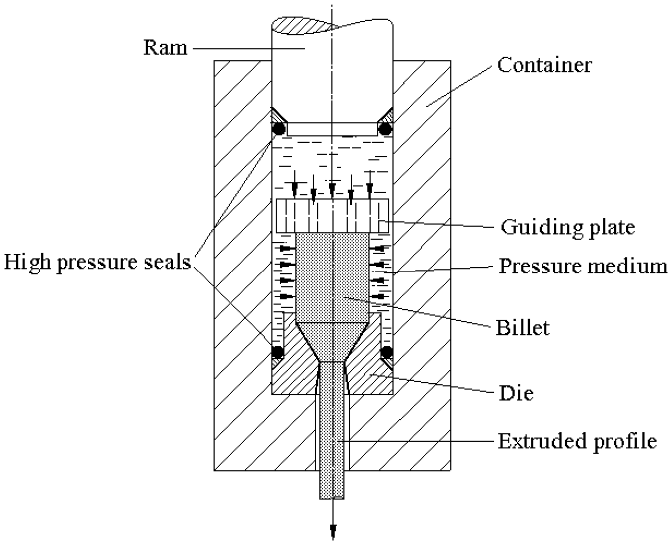


*Figure 1: Principle of hydrostatic extrusion*

- the pressure of the pressure medium has to be controlled to achieve a constant profile exit speed
- the extrusion ratio can be extremely high (up to 300), leading to the highest true strain (up to20) and strain rates (up to 300 $s^{-1}$) of all metal forming processes
- the high extrusion ratio and the high extrusion speed cause big gradients of the simulated technological values. These big gradients cause early degeneration of the FE mesh and necessitate frequent remeshing operations. The numerical convergence of the simulation is interrupted.

## 2. MODEL OF HYDROSTATIC EXTRUSION

Based on the characteristics mentioned above a model of hydrostatic extrusion has been developed,which is shown in Figure 2.

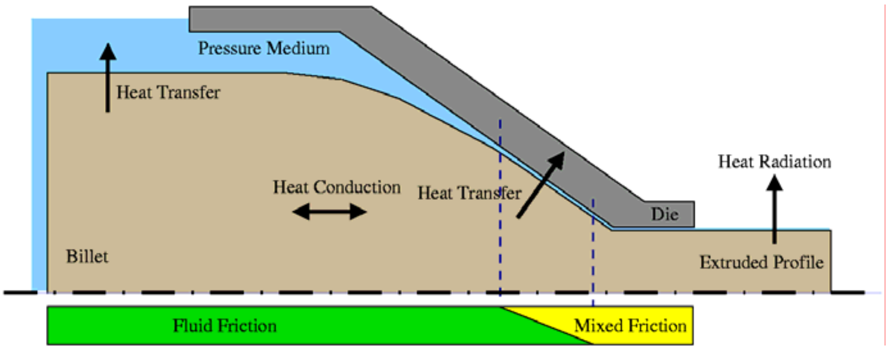


*Figure 2: Model of hydrostatic extrusion with friction conditions*

The workpiece is represented by a FE-mesh. In a thermo-mechanically coupled simulation following temperature related phenomena are calculated by the solver:

- heat generation due to dissipation during the deformation
- heat generation due to friction
- heat conduction within the workpiece
- heat transfer to the tools
- heat transfer to the pressure medium
- heat radiation at the profiles surface outside of the extrusion chamber.

The die (and for hollow profiles: the mandrel, which is not shown in Figure 2) are assumed to be rigid. Elastic deformation is neglected. In the simulation model, the surfaces of the tools are represented by a FE-mesh. The friction between the tools and the workpiece is modeled using the Coulomb's law of friction.

The pressure medium applies its pressure on the workpiece within the geometrical limits of the pressure medium. The penetration depth of the pressure medium is represented by the right dotted line in Figure 2. The pressure medium is viscous and has a low speed relative to the workpiece. Consequently, there is no friction between the billet and the pressure medium. Therefore, it is sufficient to represent the pressure medium in the simulation model as a boundary condition of the workpiece, modeling only its pressure properties. In the area where the pressure medium penetrates into the gap between the die and the billet, a linear transition between the application of the forming force on the workpiece takes place: an increasing fraction of the forming force is transfered by the tools and an decreasing fraction of the forming force by the pressure medium. Consequently, the friction condition changes from fluid friction at the entrance of the die to mixed friction inside of the die. This transition takes place between the two dotted lines in Figure 2.

The simulation model of hydrostatic extrusion was developed by implementing its special features to the finite element simulation package, which consists of the programs PEP [5] and LARSTRAN/SHAPE [6]. PEP ("Programmer's environment for Pre- and Postprocessing") is a powerful Pre- and Postprocessor for finite element simulations, which was developed at the IBF. Due toits modularity, it is particularly suitable for developing the simulation model of hydrostatic extrusion. The solver of choice for developing this model is LARSTRAN/SHAPE, because it is especially suited for the simulation of processes with large strains using a "Lagrange" mesh formulation. In addition, it is capable of thermo-mechanical coupled simulations.

## 2.1. PRESSURE AND EXTRUSION SPEED CONTROL

A module to control the pressure during the simulation of the extrusion process has been implemented to the simulation model. Figure 3 shows the principle of this pressure control and its implementation into the program PEP. The pressure control calculates the actual velocity of the profilefrom the (technological) values provided by the FE simulation package. The future pressure load is calculated with a linear control function according to the control parameters, which have to be preset by the user. To increase the numerical stability of the simulation, the pressure control is damped by limiting the pressure adjustment per time step.

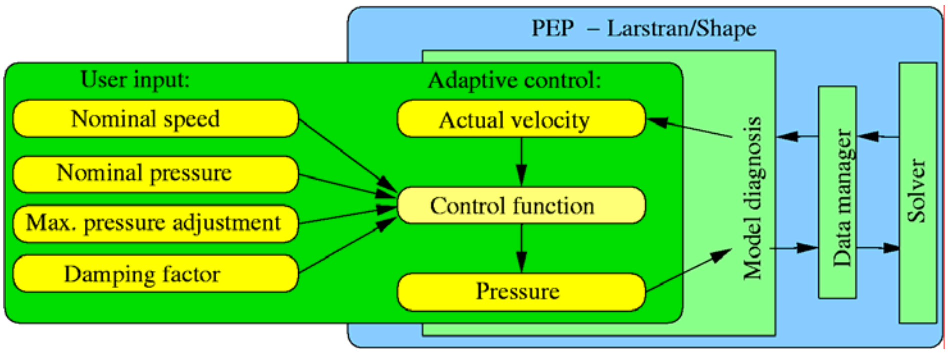


*Figure 3: Pressure control*

### 2.2. REMESHING AND MESH OPTIMISATION

The large strains of the hydrostatic extrusion process necessitate a high number of remeshing operations during the simulation and a mesh well adopted to the given geometry.

A template remeshing module was added to the simulation package to allow for the use of structured[1] FE-meshes, which minimize the calculation time and are well adopted to the given process geometry. Such a mesh is stored as a mesh template on the file storage to be reused for the remeshing operations during the simulation.

The principle of the template remeshing technique is illustrated in Figure 4: when the mesh is degenerated and a remeshing becomes necessary, the mesh template is read from the file storage and a geometrical adjustment to the actual geometry of the workpiece is performed. After transferring the technological values from the old mesh to the new mesh, the simulation can be continued.

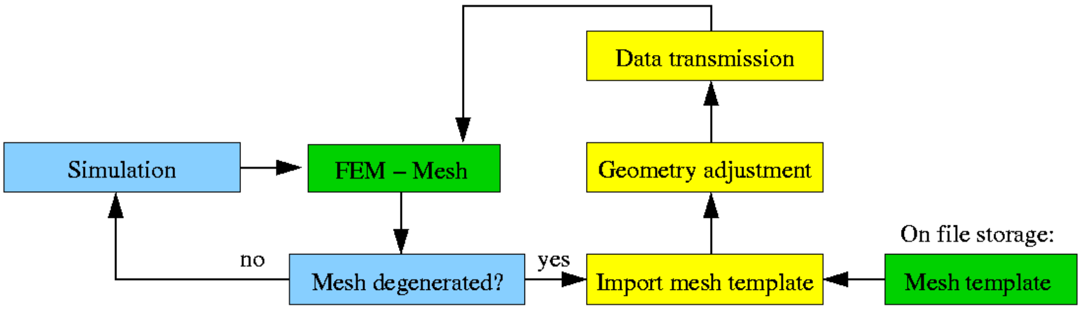


*Figure 4: Principle of template remeshing technique*

Additionally, during the geometry adjustment, the geometry of the mesh template is optimised to reduce the number of remeshing operations: The velocity field of the last time step prior the remeshing is used to deform the mesh in the opposite direction of the extrusion process as far as the element test [4] permits. Such optimised mesh templates have a low element quality, but during the subsequent simulation, the element quality is increasing, and later on, it decreases again until the mesh has to be replaced in the next remeshing operation. Figure 5 illustrates this behaviour. For comparison, the behaviour of regular remeshing operations (done by the "QuadGrid", or "Quad-Pave" algorithm) is shown: a new mesh is generated, which is of high element quality. This mesh deteriorates during the simulation until it reaches the remeshing criterion. By the use of the opti-mised net templates the need for remeshing operations is cut by almost 50%.

[1] A "structured" mesh conforms to certain geometrical rules, and its nodes are numbered in a special order. This results in a smallest possible bandwidth of the mesh.

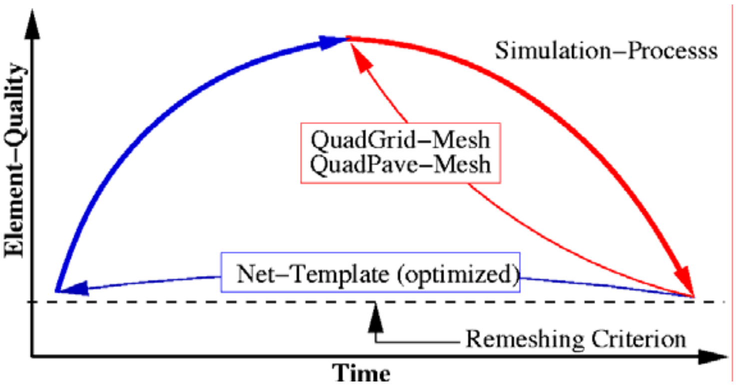
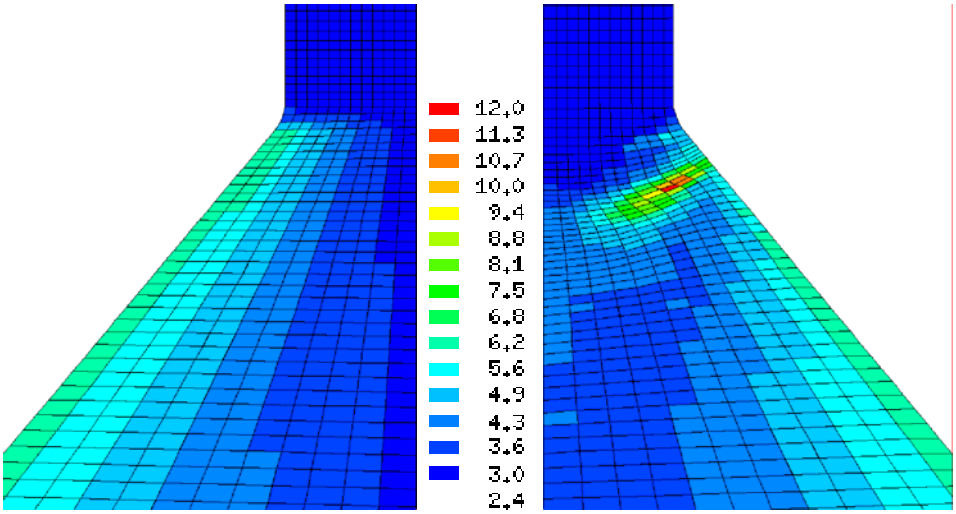


*Figure 5: Element quality during simulation and remeshing*

Figure 6 shows the element quality of a mesh template before and after being optimized. Particulary the area just before the die exit has been deformed by the optimisation process. This is the area, in which the fastest deformation of the workpiece takes place.

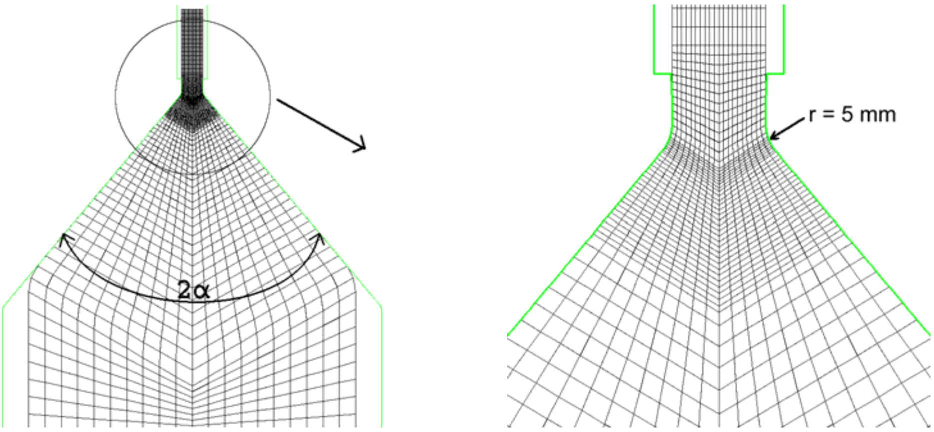


(a) not optimised (b) optimised

*Figure 6: Element quality of optimised and unoptimised mesh-templates*

## 3. VALIDATION OF THE SIMULATION MODEL

The validation of the model was done by comparison of the results of the pressure load computed with the simulation model and the pressure plot measured during the extrusions performed by the partner Boliden HME on their industrial hydrostatic extrusion press.



(a) View of deformation zone (b) Detailed view of die exit

*Figure 7: 2D-Simulation model of hydrostatic extrusion*

### 3.1. PARAMETRISATION OF THE SIMULATION MODEL

In the first stage, the simulation model was developed for the simulation of axial symmetric processes, which can be simulated by 2D-models. For the validation of this model, the simple geometry of a bar was chosen, which model is shown in Figure 7.

**The tools** are modeled as rigid, and their elastic deformation is neglected. Therefore only the contact area between the tools and the workpiece needs to be modeled for the numerical simulation. The die geometry is shown in Figure 7. The die exit radius was increased to r = 5 mm in order toallow for coarser discretisation of the workpiece. The diameter of the extruded profile is 10 mm. The cone angle is $2 \cdot \alpha = 80°$. The die has a temperature, which is assumed to be constant $\upsilon$DIE= 50°C during the entire process.

**The work piece** is the billet, which becomes the extruded profile during the process. To shorten the simulation time, the billets' geometry was shortened for the simulation from the original 800 mm to 300 mm, which is sufficient for reaching the steady state in the simulation. This shortening did not influence the results of the simulation, because the steady state of the extrusion process is reached prior to the extrusion of such lengths. Additionally, the tip of the billet was prolonged, so that it already penetrates the die at the start of the simulation process. This can be done, if the simulated results are of interest when the steady state of the extrusion process is reached and not the initial diefilling

**The flow curve** properties are the most important boundary conditions for the modeling of theworkpiece material. The flow curves have been measured with the upsetting test, up to a true strain of $\varphi_v = 0.7$ in the temperature range of 250 to 350°C. The flow stress for a true strain $\varphi_v$

exceeding the measured range is extrapolated using the last measured value. This extrapolation is justifiable because of the material recrystallising during the extrusion process.

The material's properties for elasticity (Young's modulus), heat capacity, thermal expansion and heat conductivity within the workpiece are approximated by the values as a function of the temperature of pure magnesium published in [2].

The friction coefficient between the workpiece and the pressure medium (castor oil) is assumed to be $\mu = 0$ and $\mu = 0.1$ between the workpiece and the die.

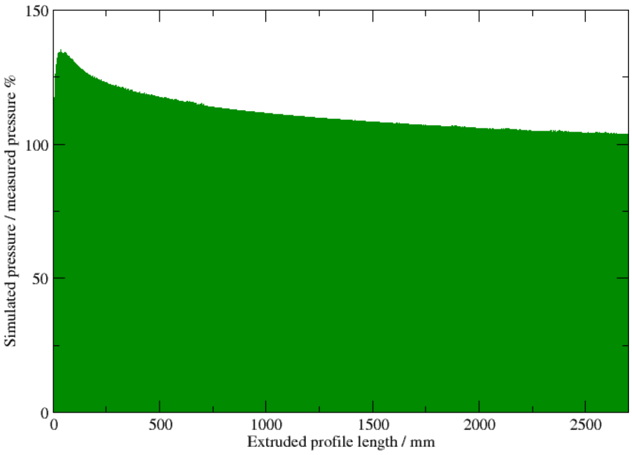


*Figure 8: Plot of the simulates extrusion pressure in percent of the measured steady state extrusion pressure*

**Process properties:** the temperature of the pressure medium is assumed to be constant $\vartheta = 50°C$ during the entire process. The pressure medium maximum penetration depth into the die (rightdotted line in Figure 2) is 35mm before the die exit. The width of the pressure reduction area (between the two dotted lines) is also 35 mm. The pressure load control is set to an extrusion speed of v = 900mm/s.

## 3.2. RESULTS

During the transport of the preheated billet from the furnace to the extrusion press, the billet cools due to radiation. Also, when the press chamber is filled with the pressure medium, the billet becomes cooler. Since the flow curve of the material is dependent on the materials' temperature, the temperature of the billet at the beginning of the extrusion was determined in a thermal simulation.

This cooling was simulated in two steps: 1. The transport, which takes 90 s and 2. The filling of the oil into the extrusion chamber, which also takes 90 s. The thermal properties used for the simulation have been taken from literature [3, 1]. The temperature field computed in this simulation provided the starting condition for the subsequent extrusion simulation.

In Figure 8 the ratio (in percent) between the simulated and the measured steady-state extrusion pressure is plotted as a function of time from the beginning of the simulation process. The simulated pressure converges asymptotical to the measured pressure after the initial pressure peak was passed. Even although the final state is not reached in the shown pressure plot, it is obvious, that the pressure will stay at about 105 % of the real extrusion's pressure.There are several possible reasons for the difference of 5 %:

- the extrapolated flow curve does not exactly match the real flow curve behaviour
- the die exit radius has been increased
- the friction behaviour is not exactly modeled
- the thermal properties of the billet, pressure medium and the die are not exactly known.

The simulated peak pressure is 130 % of the measured steady state pressure, whereas the measured peak pressure was only 10 % of the steady state pressure. This measured value is rather small compared to other extrusions, which have had a pressure peak reaching values up to 170 %. Onereason for the difference between the simulated and measured peak pressures is, that the simulation model starts with a workpiece, which already intrudes into the die.

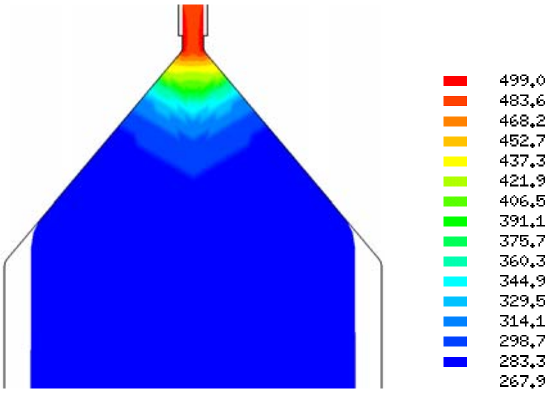


*Figure 9: Simulated temperature distribution [°C] (diameter 10 mm; alloy Mg ZM 21)*

The fact, that the simulated steady state pressure matches well the measured pressure during the further course of the extrusion conducted by HME is a good validation of the simulation model.

The temperature distribution in the deformation zone and the extruded bar is shown in Figure 9.The uniformity across the radius of the extruded product is high. Due to the high extrusion speed and therefore very short process time, the heat transfer between the billet material and the tools / the pressure medium is far lower than the heat generated due to the deformation and the friction with the tools.

The temperature distribution is the most interesting output of the simulation for the process optimization, since the hot shortness sometimes observed in the industrial process is directly related to the temperature.

The distribution of true strain in the deformation zone is shown in Figure 10. The maximum values for the true strain of 9.3 are reached at the surface of the extruded bar. The center of the extruded bar reaches true strain values of 5.5.
The strain rate, which is shown in a detailed view of the die exit in Figure 11, is highest close to the die exit radius, reaching values as high as 200s$^{-1}$.

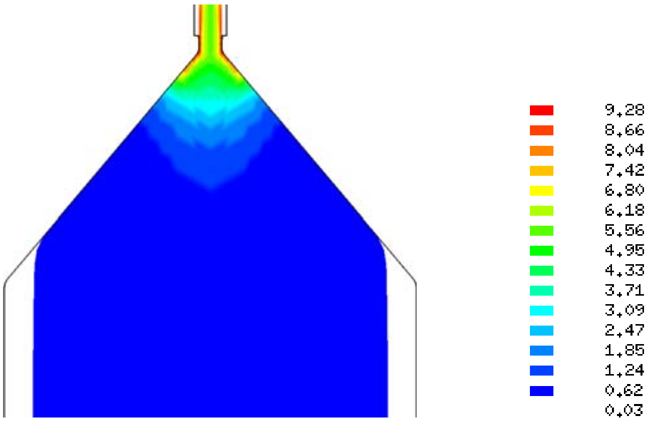


*Figure 10: Simulated true strain distribution [-] (diameter 10 mm; alloy Mg ZM 21)*

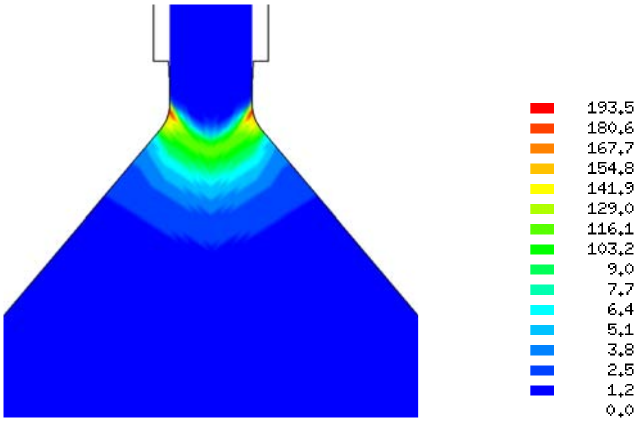


*Figure 11: Simulated strain rate distribution [1/s] at the die exit (diameter 10 mm; alloy Mg ZM 21)*

## 4. CONCLUSIONS & OUTLOOK

This article shows the modeling and first results of the finite element simulation model developed for the numerical simulation of the hydrostatic extrusion process. Focus of the work was to deal with the specific process geometry and to develop appropriate (re-)meshing techniques to avoid unacceptably long computing times. The simulation model for hydrostatic extrusion presented in this work is well capable to simulate the extrusion of axially symmetric profiles, such as bars and pipes.

The work presented here is the starting base for the development of a 3D-simulation model with high accuracy, which is necessary for the simulation of geometries such as L-shapes, square hollowprofiles and more complex geometries. The main limiting factor for the simulation of 3D-geometries, such as L-profiles and more complex geometries, is the excessively long computing time resulting from the extremely high extrusion ratio implied to the hydrostatic extrusion process. One promising possibility to cut down the computing time is the use of structured meshes for the simulation of 3D-geometries, as it is done for 2D-geometries in the work presented. The generation of those meshes is being developed.

This work shows also, that it is highly desirable to research methods to measure and describe the flow curve properties for true strains exceeding $\varphi > 0.8$ and strain rates exceeding $.\varphi > 200s^{-1}$, since the extrapolation used in this work can only be a first approximation. The development of such a method is beyond the scope of this work. A method for the measurement of flow curves at high strain rates up to $\varphi = 1000\ s^{-1}$ and a true strain of $\varphi = 0.8$ is proposed in [7].

The authors gratefully acknowledge the financial support by the European Commission within the 5th framework (GROWTH, contract number G5RD-CT2000-00424).

## 5. REFERENCES

[1] A.J. Fontana, B. Wacker, C.S. Campbell & G.S. Campbell. Simultaneous thermal conductivity, thermal resistivity, and thermal diffusivity measurement of selected foods and soils. Presentation at the 2001 ASAE Annual International Meeting, July 2001.

[2] C. Kammer. Magnesium Taschenbuch, chapter Eigenschaften von reinem Magnesium, pages77-97. Aluminium Verlag, Düsseldorf, 2000.

[3] H. Kuchling. Taschenbuch Physik. Verlag Harri Deutsch, 5. edition, 1984.

[4] M. Franzke. Zielgrössenadaptierte Netzdiagnose und -generierung zur Anwendung der Finite Element Methode in der Umformtechnik. PhD thesis, Institut für Bildsame Formgebung, RWTH-Aachen, 1999.

[5] M. Franzke. PEP Programer's Environment for Pre-/Postprocessing - Handbuch zur Version 3.30. IBF, RWTH-Aachen, 2001.

[6] R. Diez, U. Hindelang, A. Kurz. Revised form of LARSTRAN 80 documentation. LASSOIngenieurgesellschaft, January 1996.

[7] R. Kopp, T. Rehrmann. Enhancements in Determining High Speed Flow-Stress Data by the useof Compression Tests. WGP Production Engineering, X/2:19-24, 2003.

# FE MODELIRANJE PROCESA HIDROSTATIČKOG ISTISKIVANJA (EKSTRUZIJE) MAGNEZIJUMA

*R. Kopp, G. Barton*

## REZIME

*Hidrostratička ekstruzija je proces deformisanja koji je okarakterisan visokim odnosom istiskivanja, čak i u slučaju istiskivanja teško obradivih materijala, i to zbog postojanja visokih hidrostatičnih pritisaka u procesu. Takvo naponsko stanje poboljšava obradivost i (ako je proces kombinovan dobrim podmazivanjem) homogenost mehaničkih osobina po zapremini radnog komada.*

*Simulacioni model hidrostatičke ekstruzije kreiran je i primenjen u FE paketu PEP&LASTRAN/SHNAPE. Kako bi se na zadovoljavajući način respektovala činjenica da se proces izvodi u uslovima visokih odnosa redukcije, "remeshing" tehnika je primenjena u simulaciji.*

*Kreirana simulacija toka materijala omogućava određivanje brzine deformacije i raspored napona i temperature po zapremini radnog komada. Model je verifikovan upoređivanjem dobijenih veličina u procesu simulacije sa eksperimentalno dobijenim veličinama.*

*Rad predstavlja početni korak za razvoj 3D – simulacionog modela sa velikom tačnošću, (npr. za "L" profil, šuplji kvadratni profil i druge kompleksne geometrije). Glavni ograničavajući faktor za 3D simulaciju je veoma dugo vreme procesiranja podataka.*

*Fokus rada je usmeren pre svega na specifičnost geometrije, na razvoj odgovarajućeg postupka "remeshing-a". Kreirani model omogućuje i simulaciju axi-simetričnih profila (šipke, cevi).*

*Rad takođe upućuje na potrebu razvoja metoda za određivanje krivih tečenja za deformacije* $\varphi_e > 0{,}8$ *i brzine deformacije* $\dot{\varphi} > 200\ \mathrm{s}^{-1}$ *jer ekstrapolacije, koje su korišćene u ovom radu, predstavljaju samo prvu aproksimaciju.*

*Razvoj takvih metoda je bio van ciljeva ovog rada.*