Development and industrial verification of QForm-Extrusion program for simulation of profile extrusion

S. Stebunov[1]*, A. Lishnij[1], N. Biba[2], S.Bellotti[3], P.Bevilacqua Fazzini[3]
[1]QuantorForm Ltd., 121609 P.O. Box 39, Moscow, Russia, info@qform3d.com
[2]Micas Simulation Ltd., PO BOX 4190, Stourport On Severn, Worcestershire, DY13 0WU, United Kingdom, nick@qform3d.com
[3]COMPES S.p.a., Via Castegnato, 6/C - 25050
Rodengo Saiano Brescia, Italy, stefano.bellotti@compes.it
* Corresponding Author

INTRODUCTION

QForm-Extrusion is a special-purpose program for aluminium profile extrusion simulation newly developed by QuantorForm Ltd. The software includes two discreet models. The Lagrange model is for transient simulation at the first stage of the process while Lagrange-Euler model is for simulation at a steady state stage [1, 2]. At the first stage the finite element mesh follows the material flow and by these means precisely traces the progress of the die filling. In a case of the hollow profiles the material flow separates at the “bridges” that support the mandrel and then merges again in welding zones and shows the details of the material flow very clearly. The simulation of the transient stage of the extrusion is by means of the Lagrange model which goes quite quickly at the beginning but then slows down when the material reaches the die orifice where further simulation with this model becomes ineffective. The Lagrange-Euler model is based on the assumption that the tool set is already completely filled and the domain of the material flow in the inside of the tool does not change. Thus the finite element mesh in the inside of the tool represents space domain subject to simulation. This means that the mesh here is immovable while the material flows through it. The advantages and drawbacks of this method were analysed in monograph [3] where different types of the elements where used to get the solution. This approach allows the program not to remesh the domain inside of the tools but just to calculate the velocity in the nodes within it. On the other hand after the orifice we have the free end of the profile that increases its length very quickly. Due to non-uniform material flow the profile that leaves the orifice may bend, twist or buckle. The simulation goal is to predict this undesirable shape deterioration and to find ways to minimize it. To simulate this stage of the extrusion process both the Lagrange model and the Lagrange-Euler models are coupled together in QForm. Validation of the model was performed for the load prediction, the material flow pattern and the temperature distribution using special model experiments and industrial practice. The developed approach for profile extrusion simulation has shown good results at the Benchmark test in Bologna [4].

SIMULATION OF EXTRUSION ON THE BASIS OF LAGRANGE-EULER APPROACH

From practical point of view the most important is quasi steady-state stage when the product shape and properties are formed. During quasi steady-state stage of the process some parameters (the temperature, the load) may vary but this variation does not influence material flow considerably and can be neglected. Generally the source data for simulation include:

- The geometric models of the tools originally created in some CAD system and then converted into finite-element representation.
- The properties of the extruded material (the flow stress and thermal properties)
- The conditions on the contact surface of the extruded material with the tools (the friction, the heat transfer coefficient, the temperature of the tools).
- The process parameters (the initial temperature of the billet, the extrusion speed, the pulling force).

QForm-Extrusion is the finite-element simulation program specially developed for modeling and optimisation of the profile extrusion. Due to big model size it is 64-bit application that allows dealing with up to 500 000 of nodes on PCs with up to 16 GB of RAM and it runs the simulation in parallel on up to 8 CPUs. The program includes special module QShape to import the CAD models of the extrusion tools and to generate FE mesh inside the tools and to create the simulation domain. Within QShape there is a special module Bearing Editor that allows to modify bearing design, i.e. the bearing length and choke angle along the bearing evolute. The mesh generation in the program is completely automated and does not require any user's interference. The program also includes pre- and post-processors and provides different kinds of results visualization and analysis.

MESH GENERATION IN THE DOMAIN

Proper setting of the tooling geometry is vital to get successful simulation. As well known from extrusion practice there are three basic types of extrusion dies: solid dies, semihollow dies, and hollow dies, which produce solid profiles, semihollow profiles, and hollow profiles, respectively. Meanwhile for simulation purposes we distinguish the types of the die design in slightly different way (Figure 1):

1. Flat dies for making solid profiles
2. Hollow and semihollow dies with flat die surface.
3. The dies that have non-flat die surface

The simulation of extrusion process is performed within so called "simulation domain". It is the volume of the extruded material that fills the container and the inner space of the die up to the exit from the bearing land. The domain is to be created using the models of the tools. In general the domain can be created in three different ways:

1. For flat solid dies it can be created using two-dimensional drawings of the profile, the chamber and the container. Such the domain initially has the bearing of constant length that later can be modified using Bearing Editor.
2. All 3D solid models of the tooling set as are to be preliminary merged into a single body in a CAD system as shown on Figure 2a. Then the domain is to be created by means of import of this 3D die geometry from CAD system into QShape and subsequent conversion into finite element representation. Then QShape creates the domain as shown on Figure 2b, 2c.
3. The domain can be created using the tools as separate parts created in CAD and then joined together directly in QShape program.

Currently the second option for the domain generation is realised in the program while the other options are to be added soon. The most critical stage of the domain creation is automatic recognition of the bearing zone and converting this part of the die design into parametric representation. Parametric representation of the bearing is necessary for its interactive modification without going back to CAD system. After such parametrisation the geometry of the bearing is represented not by finite elements but by means of splines and can be easily modified in the program. Particularly it is possible to change the bearing length and choke angle in the way to provide desired material flow. It is also important to have parametric representation of the bearing when we deal with the dies for producing several similar profiles in a time and any modification of the bearing of one of them must be automatically repeated for all the other profiles. The last stage is generation of the finite element mesh inside of the domain. As soon as the domain was created the program automatically sets proper boundary conditions over its surface and provides simulation of extrusion process to the profile length specified by the user (Figure 3). It is important to mention here that described sequential stages of the simulation domain creation can be automated and an interface to the other CAD/CAM programs for die design and manufacturing can be developed on request.

The mesh inside the domain is built using tetrahedral elements while when the profile leaves the die orifice it is approximated by prism elements. The quality of the finite element mesh is critical to obtain accurate results. Insufficient mesh density or its too big gradients may cause non-convergence problem and deteriorate the quality of the simulation (Figure 4). It is especially critical if the mesh has improper density distribution at the entrance into the bearing where the most intensive deformation takes place. While in the extruded profile when it leaves the die it is enough to have 2-3 elements across the profile, in the deformation zone it is necessary to have not less than 10 elements layers. Thus the finite element mesh is to be created iteratively adapting it to the solution behaviour such as the velocity gradients at the entrance to the die orifice.

BEARING EDITOR

Bearing Editor is a part of QShape program and it is intended for interactive modification of bearing design. To show how it can be used let us consider the simulation of extrusion of hollow angular profile. The first attempt of the simulation has shown the same deformation of the front tip of the profile as in real extrusion (Figure 5). The reason of such shape deterioration is much lower metal velocity in the "corner" of the profile that in its legs. Bearing Editor displays the evolutes of the bearing length of all contours of the profiles (outer and inner ones) and shows them as graphs (Figure 6). Moreover after completing of the simulation Bearing Editor displays the evolute of the velocity profile on the same graph as the evolute of the bearing length that allows to see how they correspond each other (Figure 7). For this particular angular profile the initial reduction of the bearing length in corner area of the profile was not enough to equalise the velocity (Figure 8b). With its further reduction the velocity profile becomes uniform as well as the front end of the profile when it leaves the die orifice (Figure 8b).

MODEL VERIFICATION BY SIMULATION OF INDUSTRIAL CASES

Industrial verification of the program was done using wide range of different solid and hollow profiles of different complexity with various extrusion ratio that are produced by COMPES S.p.a. Totally more than 20 profiles where investigated. It is impossible to measure the velocity distribution along the profile contour in real extrusion thus the only way is to compare the shape of front tip of extruded real profile with the shape of its front end predicted in simulation. There were several goals of such industrial investigation:

1. Testing and improvement of the methods of the geometry data transfer from industrial system of the die design into simulation program.
2. Estimation of the accuracy of the simulation.
3. Use the results of the tests for further development of the numerical model and the software.

The results obtained in these tests were divided into three groups. The first group shows very good correspondence between simulation and real extrusion like, for example, for the profiles No1 and No2 in Table 1. Some profiles have shown not that exact but still acceptable correspondence like profile No3. Some profiles have shown bad correspondence between simulation results and real profile like profile No4.

These results were analysed and the reason of inaccuracy was found in insufficient mesh density in the zones of the most intensive deformation of the material at the entrance to the bearing zone. When the program was supplied with additional optimization of the mesh density the correspondence of the simulation results to real extrusion was significantly improved as seen in the second simulation for the profile No4 (Table 1). On the other hand such increasing of the mesh density causes extending of the overall problem size that essentially requires hundreds of thousands of nodes for industrially produced profiles of complicated shape. To fit this demand QForm-Extrusion is provided with highly efficient facilities for mesh generation and solving of the system of equations and uses parallel computation technique to speed up the simulation.

CONCLUSIONS

- QForm-Extrusion is a special-purpose finite-element simulation program based on Lagrange-Euler approach for the use in industrial environment.
- The program has the module for data import from CAD systems that allows interactive modification of bearing design and that also can be used for bearing design optimisation.
- The program provides automated simulation of the material flow through the die orifice and shows the shape of the front end of the profile that can be compared with front tip in real extrusion.
- Industrial tests of the program have given vital information on the effectiveness of this approach and have provided the data for further development of the model.
- After improvement of the algorithms the program has shown very good correspondence of simulation results with real extrusion for wide range of the profiles that means it can be successfully used for die design development and extrusion technology optimisation in industry.

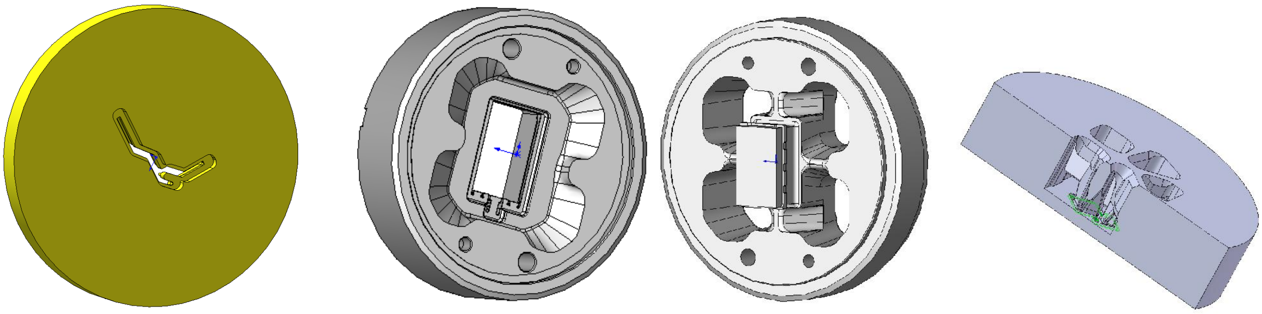
REFERENCES

[1] Biba N., Stebunov S., Vlasov A., "Application of QForm Program for Improvement of the Die Design and Profile Extrusion Technology", 2008, Proceedings of the Ninth International Aluminum Extrusion Technology Seminar & Exposition, Orlando, USA.

[2] Stebunov S., Biba N., Lishnij A., "QForm-Extrusion – the program for simulation and development of profile extrusion technology", 2008, Proceeding of the Conference on Aluminium in Building, Moscow, pp. 82-83

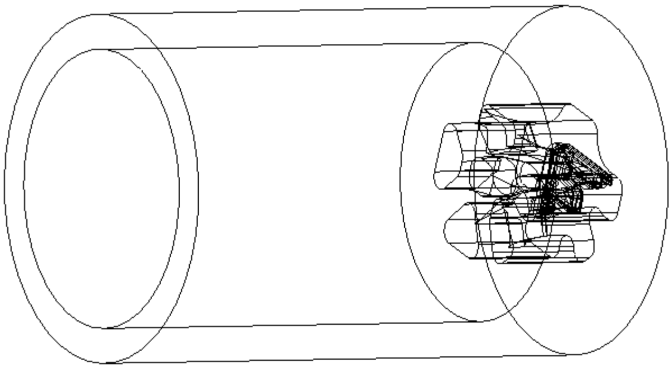
[3] Danchenko V.M., Mileninn A.A., Golovko O.M. "Production of profiles from aluminum alloys. Theory and technology", 2002, Dnepropetrovsk, Ukraine, System technologies (In Russian).

[4] Donati L., Tomesani L., Schikorra M., Tekkaya E., 2007, "Extrusion Benchmark 2007". Proceedings of the Conference Latest Advances in Extrusion Technology and Simulation in Europe. Bologna, pp. 89-95

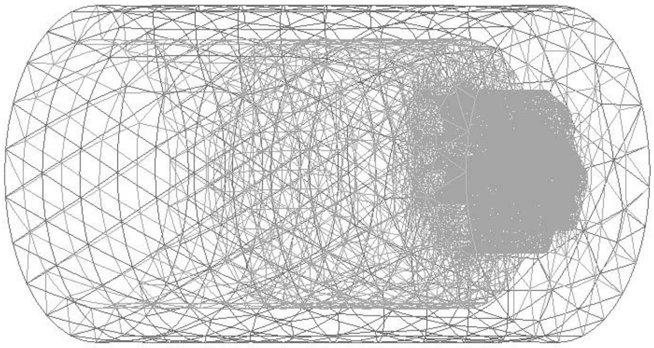


(a) (b) (c)

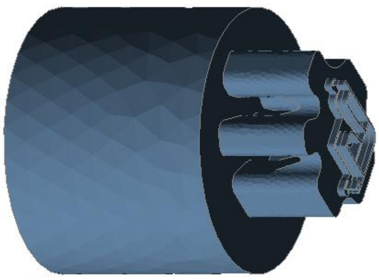
Figure 1: Different types of the dies for extrusion of Al profiles: (a) flat die for solid profiles; (b) mandrel die for hollow profiles; (c) die with stepped chamber of bearing



a.



b.



c.

Figure 2: Die set merged with the container to form a single solid body for subsequent creation of the simulation domain inside it: (a) CAD model; (b) finite element model of the tooling set; (c) simulation domain automatically generated inside of the tooling set.

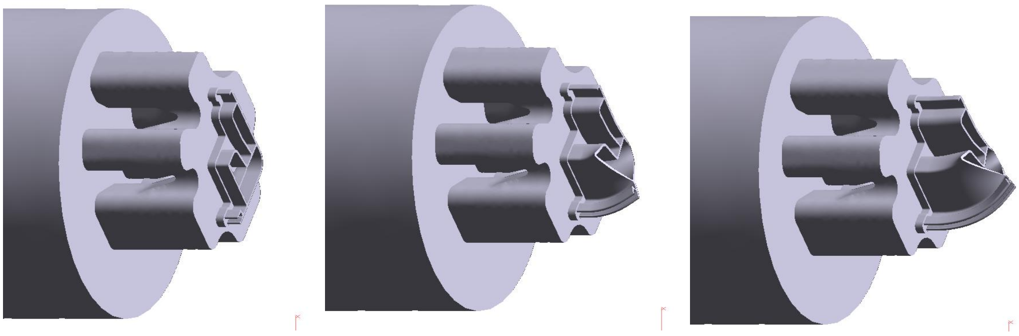
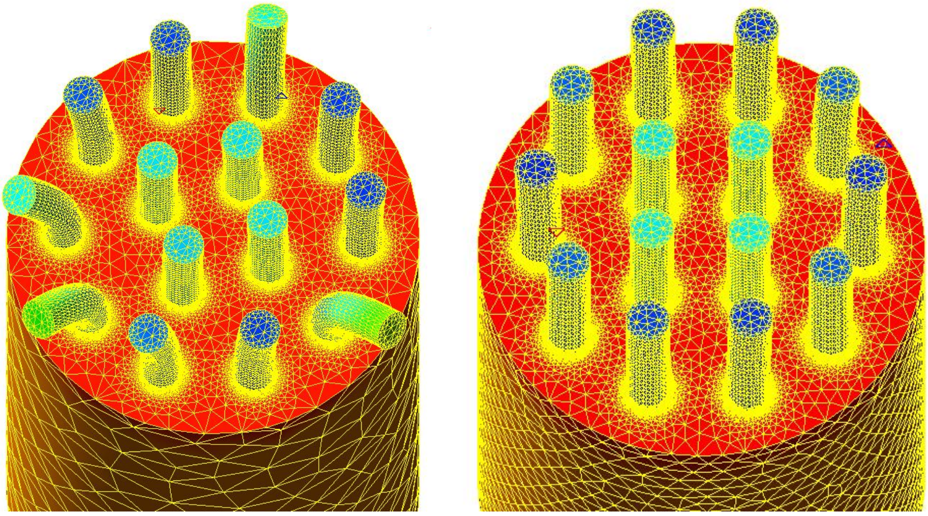
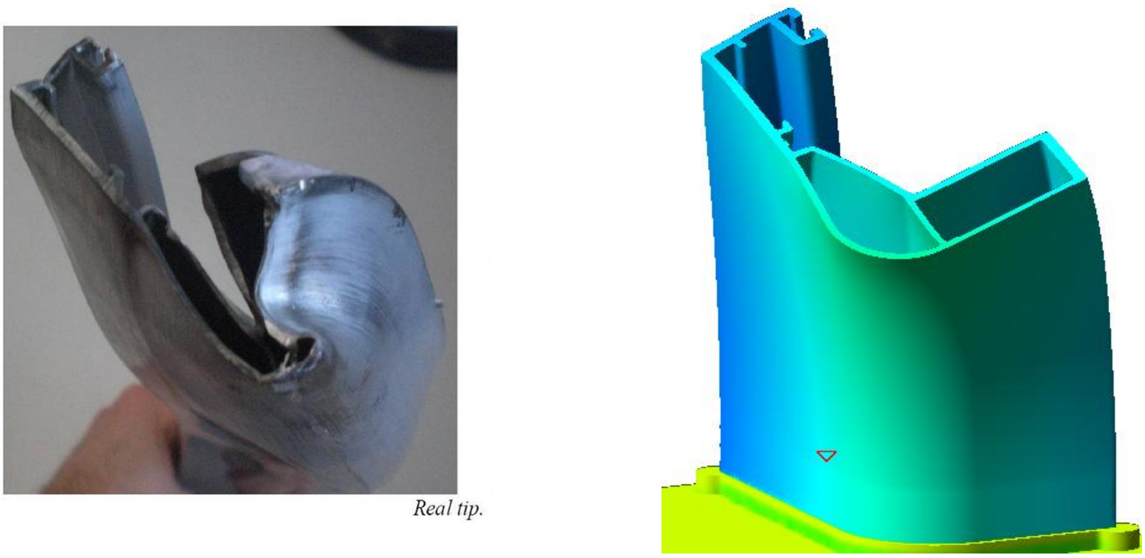


Figure 3: Several steps of profile extrusion simulation using the simulation domain shown on Figure 1.



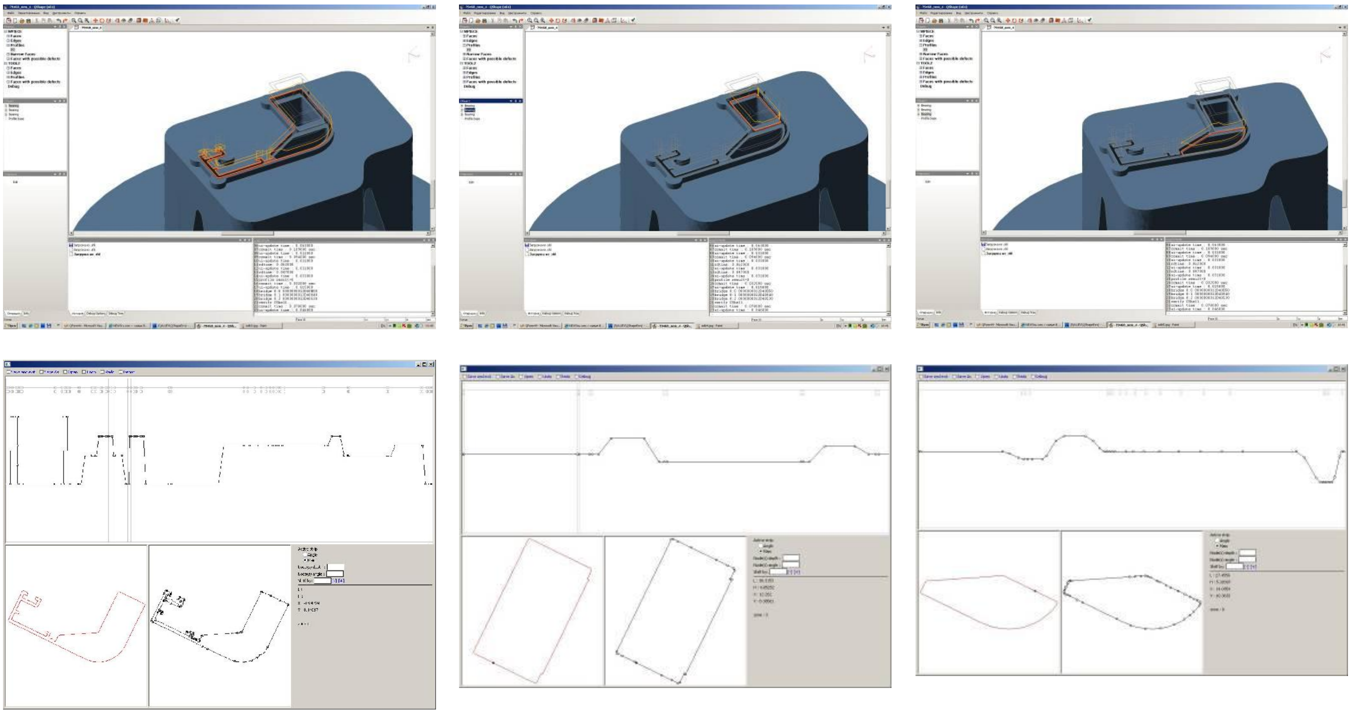
(a) (b)

Figure 4: The influence of the finite element mesh quality on the material flow: (a) not sufficient mesh adaptation causes fluctuation of the material flow; (b) optimal adaptation provides accurate simulation.



(a) (b)

Figure 5: Comparison of real front tip (a) and its shape obtained by the simulation (b) for hollow angular profile.



a. b. c.

Figure 6: The evolutes of outer (a) and inner (b, c) bearings of hollow angular profile displayed in Bearing Editor.

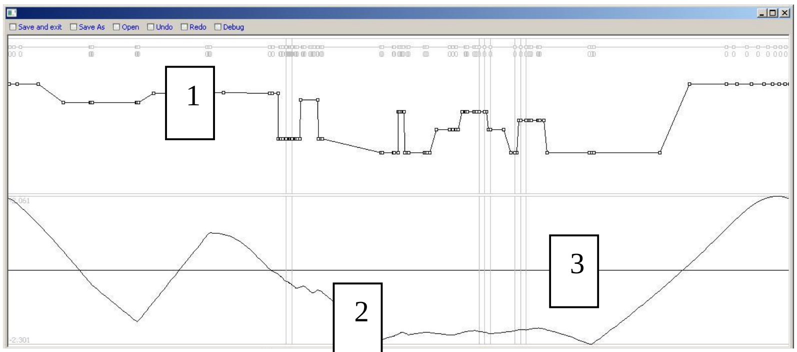
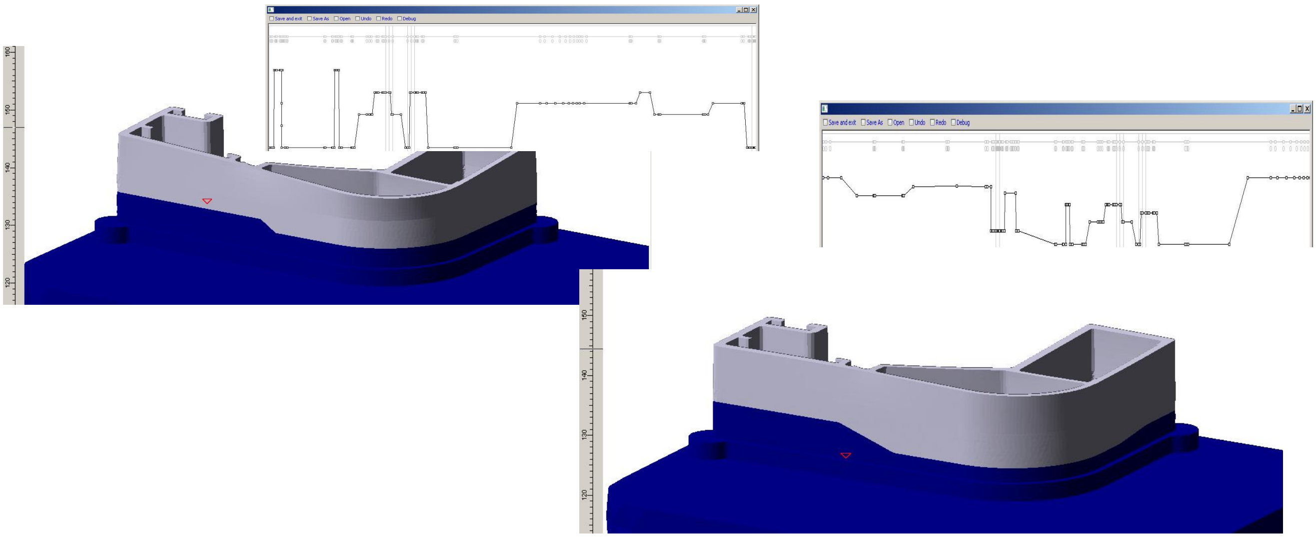


Figure 7: The evolutes of the bearing length (1), the profile velocity distribution (2) and average profile velocity shown as a line (3)



a. b.

Figure 8: Two subsequent stages of bearing length optimization: (a) not sufficient variation of the bearing length; (b) bearing length variation provides uniform velocity distribution.

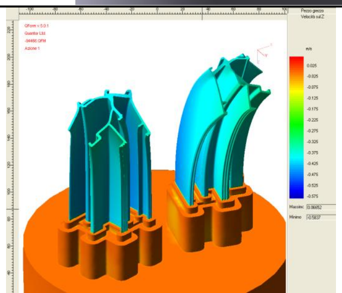
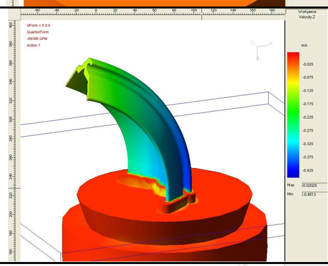
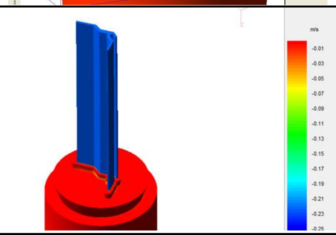
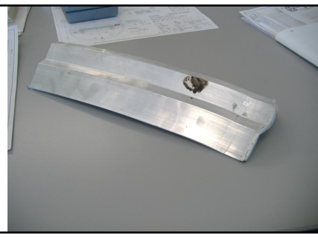
Profile number	Results evaluation	Simulation	Photo of the profile tip
1	Good correspondence between simulation and experiment		
2	Good correspondence between simulation and experiment		
3	Acceptable correspondence between simulation and experiment		
4	Bad correspondence due to insufficient quality of the mesh		
	Good correspondence after improving of the mesh density distribution		

Table 1: Some examples of industrial tests for different profiles.