Semana de Engenharia 2010
Guimarães, 11 a 15 de Outubro

COMPUTER AIDED DESIGN OF EXTRUSION DIES FOR COMPLEX GEOMETRY PROFILES

Nelson Gonçalves
Department of Polymer Engineering
E-mail: nelson.goncalves@dep.uminho.pt

KEYWORDS
Computational Fluid Dynamics

ABSTRACT
This project aims at further generalise the concept of automatic design of profile extrusion dies initiated by the research group some years ago, in order to be able to deal with a wider range of profile geometries. The design code developed so far carries out an automatic search of a final geometry via an optimisation routine coupled with geometry and mesh generators and a 3D computational fluid dynamics (CFD) code based on the finite volume method (FVM).

The CFD code is able to model the flow of polymer melts in channels, but it is not adequate to deal with complex geometries since it is limited to structured meshes. This PhD programme aims at developing a new computational rheology code, based on the same numerical method (FVM) but using unstructured meshes, which are required to model the flow in more complex geometries. Similarly to what was done with the existing code (using structured grids) this new code will be integrated in an optimisation methodology, which will enable to automatically search the best geometry of the flow channel for the production of a specific extruded profile.

NUMERICAL CODE DEVELOPMENT

The development of the numerical code started with the resolution of the General Conservation Equation, which has the form:

$$\frac{\partial}{\partial t}(\rho\phi)+\nabla\cdot\left(\rho\vec{U}\phi\right)=\nabla\cdot(\Gamma\nabla\phi)+S_{\phi} \qquad (1)$$

Using the adequate variables, this equation can take the form of the Energy Conservation Equation, that allow to compute the temperature distribution in general problems, given by:

$$\frac{\partial}{\partial t}(\rho c_p T)+\nabla\cdot\left(\rho c_p\vec{U}T\right)=\nabla\cdot(\Gamma\nabla T)+S_T \qquad (2)$$

where the unknown variable is the temperature, T. The Finite Volume Method starts with the integration of all terms of equation along the time and volume. Dividing the system domain into several control volumes and forcing the conservation in each of these volumes, it is possible to obtain a equation for each volume. Using the Divergence Theorem, in steady state problems, the general conservation equation becomes:

$$\sum_f A_f \rho c_p \vec{U}\cdot\vec{n_f}T=\sum_f A_f(\Gamma\nabla T)_f\cdot\vec{n_f}+\Delta V S_T \qquad (3)$$

In interior faces, i.e. a face between two control volumes, the diffusive term $\sum_f A_f(\Gamma\nabla T)_f\cdot\vec{n_f}$ can be approximated by

$$\sum_f A_f\Gamma\left(\left(\overline{\nabla T}\right)_f+\frac{T_{nb}-T_p}{\left\|\vec{\xi}\right\|}\cdot\vec{e_\xi}-\left(\left(\overline{\nabla T}\right)_f\cdot\vec{e_\xi}\right)\vec{e_\xi}\right)\cdot\vec{n_f} \qquad (4)$$

Where $\left(\overline{\nabla T}\right)_f$ is a estimative of the temperature gradient in face f calculated with values of last iteration of iterative process, $\vec{\xi}$ is the vector that connects the geometric centre of the two neighbour cells that share the f face and $\vec{e_\xi}$ is a unitary vector with same direction.

For the convective term $\sum_f A_f\rho c_p\vec{U}\cdot\vec{n_f}T$ other approaches have to be used, because its non-homogeneity increases with the magnitude of the velocity. To estimate T_f the best strategy is to employ TVD (Total Variation Diminishing) schemes, given by:

$$T_f=T_U+\frac{1}{2}\varphi(r)\left(T_D^*-T_P^*\right) \qquad (5)$$

with a flux limiter function $\varphi(r)$ that depends of scheme that is employed,

$$r=\frac{2(\nabla T)_U\cdot\overrightarrow{X_P X_{nb}}}{T_D-T_U}-1,\ \{U,D\}\in\{P,nb\}, \qquad (6)$$

being T^* values obtained from the last known temperature field.

To solve problems where the velocity field must be computed a SIMPLE based algorithm was implemented.

NUMERICAL CODE ASSESSMENT

To test and assess the developed numerical code several problems were solved being some of the examples presented hereafter.

The first problem tested consists of two solid domains with temperature imposed in the outside boundaries, considering two alternatives for their interface: contact resistance or perfect contact.

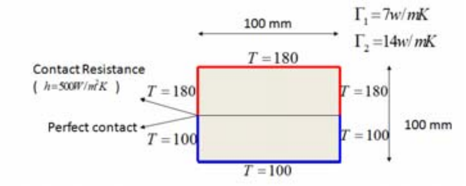

Figure 1: first problem geometry

The computed temperature distribution and the comparison with analytical solution, are both presented in the Figure 2.

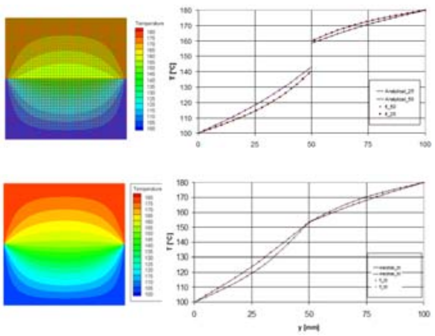

Figure 2: first problem geometry and results

The second example was used to test the flux limiter function used on convective term. In this problem a domain with the south boundary temperature set to 0 and the west boundary set 1, assuming pure convection, i.e., without diffusion. The temperature calculated by the developed code along the diagonal [AB] are presented in the graphic to compare with the analytic.

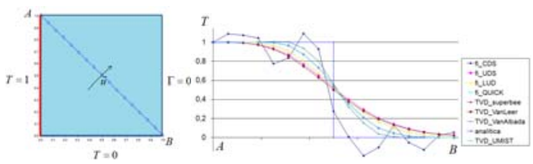

Figure 3: second problem geometry and results

The results obtained show that the best flux limiter for this problem is Van Albada.

The third example consists of a two domain problem, where a polymer layer is cooled in a metallic calibrator with input temperature imposed.

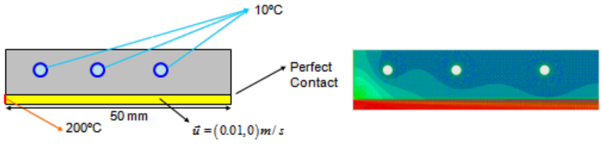

Figure 4: third problem geometry and results

The solution of the problem was compared with results obtained with alternative numerical codes.

The fourth example was used to evaluate the algorithm implemented to compute the velocity field, considering the flow of a Newtonian fluid between parallel plates. The results shown in the Figure 5 allowed the validation of the developed code, for this simple problem.

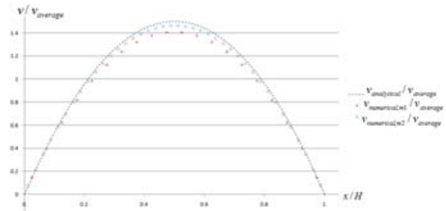

Figure 5: fourth problem results

CONCLUSION

The work performed until now in the framework of this PhD project was related to the development of a numerical code able to model the flow of polymer melts in complex geometry flow channels. Currently the code is able to solve the general conservation equation, being the algorithms required to compute the velocity field already implemented for 2D problems and Newtonian flows.

REFERENCIES

Versteeg, H. K., W. Malalasekera. 2007. “An Introduction to COMPUTATIONAL FLUID DYNAMICS - The Finite volume Method, second edition. Pearson.

Ferziger, J.H., M. Peric. 2002. Computational Fluid Dynamics. Springer.

AUTHOR BIOGRAPHIES

NELSON D. F. GONÇALVES was born in Guimarães and went to University of Aveiro, where he studied Mathematical Teaching and obtained his degree in 2005. In 2007 he obtained the Master in Mathematical Engineering in the University of Oporto. He worked for a year in a research team in University of Minho where he is PhD student. His e-mail address is nelson.goncalves@dep.uminho.pt.