A SURROGATE MODEL TO BALANCE THE FLOW DISTRIBUTION IN COMPLEX PROFILE EXTRUSION DIES

A. RAJKUMAR[1], L.L. FERRÁS[1], C. FERNANDES[1], O.S. CARNEIRO[1],
M. BECKER[2], J.M. NÓBREGA[1]
[1]*IPC/I3N – Institute for Polymers and Composites, Department of Polymer Engineering, Univeristy of Minho, Campus de Azurém, 4800-058 Guimarães, Portugal*
[2]*DHCAE Tools GmbH, Krefeld, Germany*

Keywords: *Profile extrusion, Die design, Flow distribution, OpenFOAM*

An important challenge to be addressed during the profile extrusion die design process is the difficultly in achieving a homogeneous flow distribution at the die flow channel outlet. The usual methodologies employed in the design of profile extrusion dies are based on trial-and-error procedures, and, are highly dependent on the designer's experience, leading to a high consumption of time, money and raw material.
The recent developments in computer aided die design clearly indicate the ability to accurately predict and adjust the fluid flow distribution using numerical tools. These numerical tools are a better alternative to the usual experimental trial-and-error approach, but, the number of extrusion companies using numerical tools for the extrusion die design process is small, mainly due to the lack of adequate knowledge and/or financial resources. Considering these facts, it is of significant importance to provide useful design guidelines for the extrusion die designers with no access to numerical modeling tools. Therefore, in this work we proposed some simplified design guidelines to balance the flow distribution in complex extrusion dies. The methodology circumvents the limitation inherent to analytical approaches and do not demand the use of numerical codes.
For this purpose, the profile cross section geometry is divided into simpler geometries (L and T shaped profiles), and, the flow balance on these simpler geometries is obtained by adjusting their length and thickness using a previously derived surrogate model that correlates the homogeneity of the flow to the length and thickness of the geometry. The model was obtained by an extensive and detailed numerical study, where different dimension were considered. The numerical simulations were performed for a non-isothermal Bird-Carreau inelastic fluid, using a solver implemented in OpenFOAM®computational library.
The balance of the overall flow is obtained with a new methodology, which, we assess with some case studies.