

VISUALIZATIONS AND NUMERICAL TECHNIQUES IN THE COMPLEX FLOWS ANALYSIS

Diana BROBOANĂ¹, Andreea CĂLIN², Ting OUYANG³, Cătălin M. BĂLAN⁴,
Roland KÁDÁR⁵, Cătălin MĂRCULESCU⁶, Corneliu BĂLAN⁷

Lucrarea este dedicată investigațiilor experimentale și modelării numerice a curgerilor vâscoase în geometrii complexe. Sunt studiate mișcări de tip Poiseuille, curgeri în micro-canal și mișcări de tip Couette generate de rotația unui rotor sau a unui cilindru (mișcarea Taylor). De asemenea, se analizează hidrodinamica fluidelor imiscibile: geometria interfeței în mișcarea de rotație, respectiv în mișcarea de tip Hele-Shaw, și curgerea jeturilor libere de polimeri topiți în procesul de obținere a firelor subțiri. Rezultatele evidențiază importanța deosebită pentru aplicațiile tehnice a validării codurilor numerice prin tehnici de vizualizare a curgerilor 3D în geometrii apropiate de cele reale.

The paper is concerned with the investigations of complex flows using visualization techniques and numerical simulations. The laminar motions under study are of Poiseuille and Couette types, respectively flows in 3D micro-channels and in axial-symmetric rotational geometries: (i) rotation of an impeller in close vessel and (ii) Taylor vortical flow between concentric cylinders. There are also studied flows in presence of interfaces, as rotation of immiscible liquids, evolutionary interface in a Hele-Shaw cell and the spinning of polymer melt jet in atmosphere. The work evidences a good correlation between experiments and computation and emphasis the importance in applied hydrodynamics of numerical code validation through visualizations techniques.

Keywords: Flow visualizations; numerical simulations; laminar flows; rotational motions; Taylor vortices; fiber spinning; VOF method.

1. Introduction

Numerical simulations and flow visualizations are now days well established techniques for developing studies and investigations of the flows in complex geometries and configurations. The Computational Fluid Dynamics (CFD) approach is based on the validations of numerical solutions through the

¹ Lecturer, Dept. of Hydraulics, University POLITEHNICA of Bucharest, Romania

² PhD student, Dept. of Hydraulics, University POLITEHNICA of Bucharest, Romania

³ PhD student, Dept. of Hydraulics, University POLITEHNICA of Bucharest, Romania

⁴ PhD student, Dept. of Hydraulics, University POLITEHNICA of Bucharest, Romania

⁵ PhD student, Dept. of Hydraulics, University POLITEHNICA of Bucharest, Romania

⁶ PhD student, Dept. of Hydraulics, University POLITEHNICA of Bucharest, Romania

⁷ Professor, Dept. of Hydraulics, University POLITEHNICA of Bucharest, Romania

measurements and visualizations of flows in various geometries. It is compulsory, before starting any study of the flow in a 3D geometry, to validate the numerical solutions not only for benchmark configurations, but also for fluid motions in more complex geometries (as close as possible with the real ones).

In the last period of time, the REOROM Group developed some visualization procedures in complex geometries for testing the commercial numerical cod FLUENT, which uses the Finite Volume Method (FVM) to obtain the Navier-Stokes solutions, see for details [1] and [8].

The present paper is dedicated to an overview presentation of some relevant flow visualizations and corresponding numerical solutions of the Newtonian and generalized Newtonian viscous flows, which have been investigated in the last two years by the members of the REOROM laboratory. The numerical simulations validated by visualizations constitute the base for modeling complex flow fields and optimization of various processes and geometries in which the fluid motions take place. Each of the presented cases corresponds to a practical application in fluid mechanics and/or material processing, the flows under investigations being studied in the students thesis.

The procedure to visualize the steady flow field is based on the classical streak line photography procedure, see for details [2]: small particles are introduced in the flow area and their pathlines are recorded with a high resolution Digital Cameras (the expose time and lens aperture are dependent on the velocity magnitude and the intensity of light). As reflecting particles are used either very small hydrogen bubbles (mean diameter of 20 microns), generated in the flow stream at the inlet of the tested geometry by electrolyze procedure, see Fig. 1, or alumina powder with mean diameter of 2 microns, see Fig. 2.

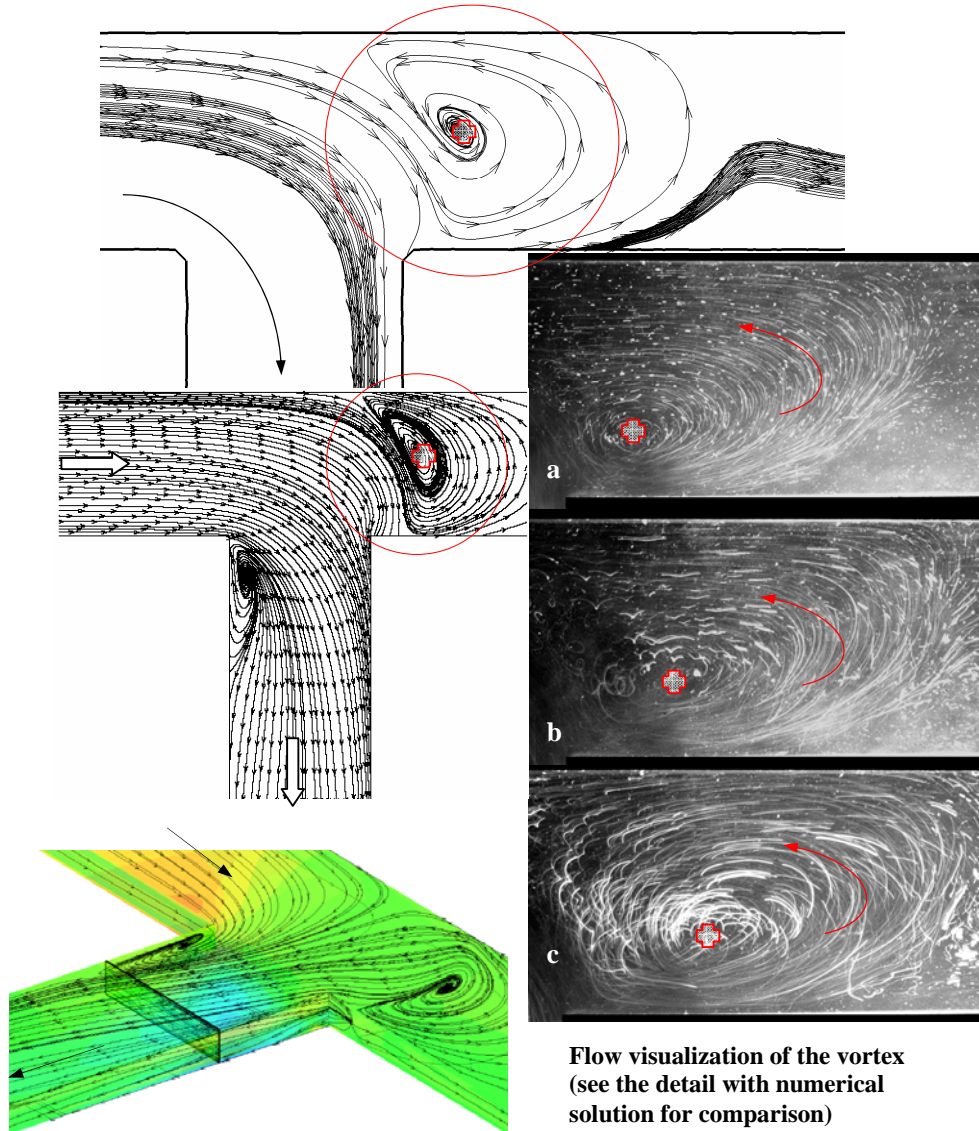
For the visualization of the interface between immiscible fluids (static or dynamic interfaces) there are used series of continuous pictures, processed later one with special image software. All numerical simulations are performed with FLUENT code, the motions in presence of a material interface being modeled with the VOF method, [3].

2. Flows visualizations and numerical simulations

The first investigated motions are of Poiseuille type, respectively the flows in channels and flows in capillaries generated by a pressure difference.

In Fig. 1 are shown the numerical simulations and visualizations of a vortical structure in a Hele-Shaw channel geometry. The differences between the experimental pictures and simulations are relevant due to the impossibility to obtain a stable 3D numerical solution at exactly the same Reynolds numbers corresponding to visualizations (the location of the “vortex center” is in reality not in the middle of the channel, as we obtain in computation). The numerical 3D

solution of Navier-Stokes equation (laminar solution) at $Re = 800$ can not fit properly the real flow which takes place at $Re = 1000$.



Numerical solutions: pathlines distributions on the middle plane of the channel.

Fig. 1. Numerical solution for a 3D flow in a “T” profile at $Re = 800$ and the corresponding visualizations of the vortex as function of the Reynolds number: a) $Re = 1000$; b) $Re = 1600$; c) $Re = 2600$. The “vortex center” in experiments is not located in the middle of the channel, as is obtained in numerical simulations (the cross section of the channel is 25 mm width and 2 mm high and the test fluid is water), see [2].

In Fig. 2 is presented the comparison between an experimental flow pattern of a viscous fluid in a micro-bifurcation and the corresponding laminar

We start the presentation with this example because some flows, especially that take place in complex 3D geometries, do not have a proper numerical solution at high Reynolds number (since the limit of the Navier-Stokes solver is unknown for a particular geometry). For that reason, it is necessary to test the flows also experimentally, in order to trust the numerical results.

In Fig. 2 is presented the comparison between an experimental flow pattern of a viscous fluid in a 3D micro-bifurcation and the corresponding laminar solution. The set-up is based on a microscopic device and a syringe pump which assures the constant flow rate into the cell. The experiments are fair consistent with the 3D numerical solutions up to $Re = 300$. A detailed investigation of the 3D capillary branches flows is presented in [4].

Another category of investigated motions are rotational flows, respectively Couette motions generated by a rotational disc (impeller) in a close vessel or by the rotation of the inner cylinder in the classical Couette-Taylor geometry.

In Fig. 3 is studied the influence of the distance between the rotated disc and the lower wall, on the vortical structures developed in a vessel, see also [5]. Figure 4 evidences the complexity of the kinematics in the same vessel, but this time the flow is generated by the rotation of an impeller. In both analyzed flows we obtain a good agreement between experiments and the axial-symmetric solutions of the Navier-Stokes equation (up to $Re = 2500$).

The Couette-Taylor flow between two concentric cylinders is one of the most investigated motions in fluid mechanics, [6]. Despite the very numerous published papers, there is still some place to evidence interesting phenomena, in relation to the transition of laminar flow to turbulence. Flow patterns visualizations of the Couette-Taylor motion - the flow between two finite concentric cylinders, where the inner cylinder is rotating and the outer cylinder is at rest - are presented in Fig. 5. The experiments were performed using a Physica MC1+ Rheometer, modified by the addition of a transparent outer cylinder. The characteristic parameters of the apparatus are: the radius ratio $\eta = R/R_i = 0.87$ and the aspect ratio $\Gamma = L/\delta = 33.53$, where R and R_i is the radius of the inner respectively the outer cylinder, L is the length of the fluid column and δ the gap. As the Taylor (Reynolds) number, respectively the angular velocity of the inner cylinder is increasing, the inertial forces outbalance the viscous forces. Therefore, the fluid suffers a transition from a one-dimensional flow, the Couette flow, to a 3-dimensional flow pattern, called Taylor vortex flow (Fig. 5.a), which consists in counter rotating toroidal vortices, called Taylor cells that travel in azimuthal direction. These vortical structures are separated in-between by purely radial flows, which play an important role in further development of the instability.

As the Taylor number is further increased, the outward radial flow intensifies leading to the axial and radial deformation of the Taylor vortices.

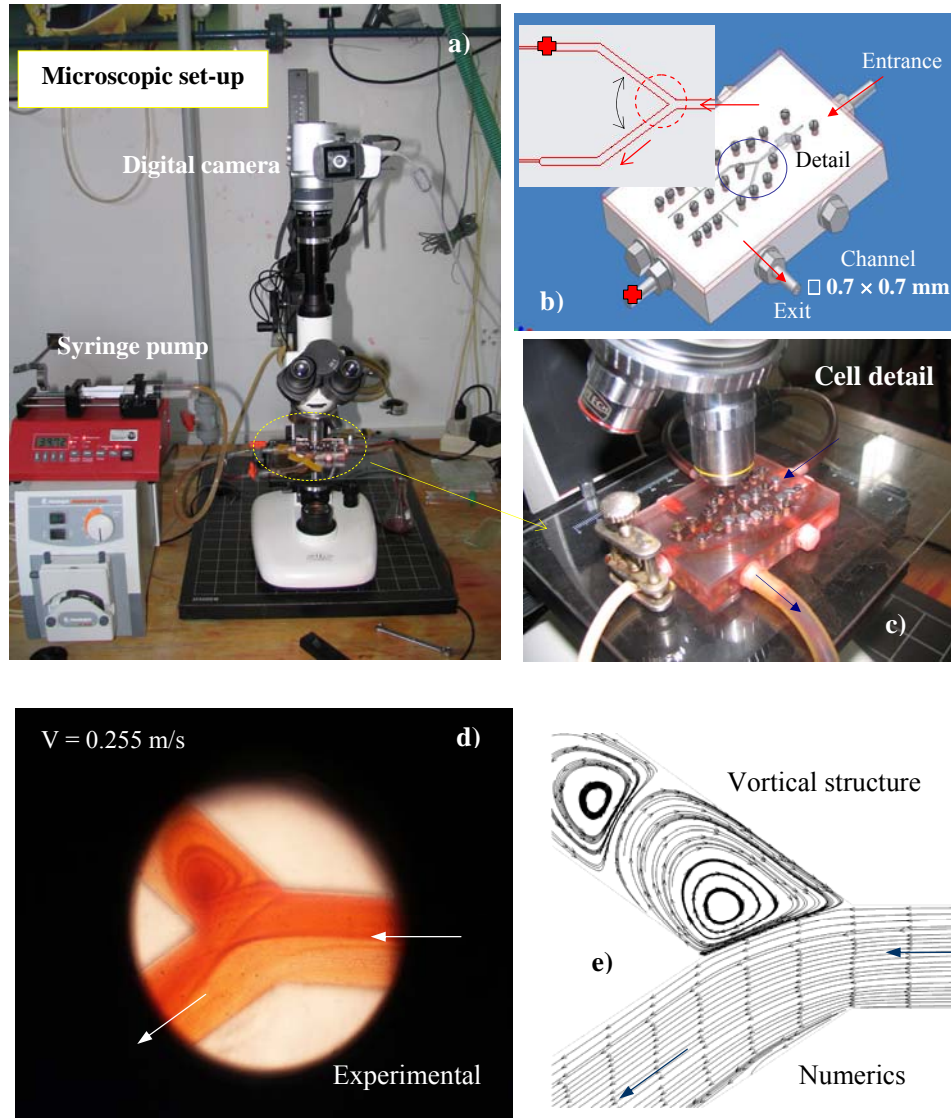


Fig. 2. Experimental set-up for visualizations of micro-flows in complex geometry (a). The fluid is transported through the measuring cell by a syringe pump and the flow pattern is recorded with a Digital Camera with high resolution. The cell is a micro-bifurcation with the nominal dimension of 0.7 mm (b, c). The picture evidences the formation of the vortex in the close branch of bifurcation (d). Same flow pattern is obtained by numerical simulations with the FLUENT code (e). The tested fluid is Newtonian (water) and the corresponding Reynolds number is $Re = 180$.

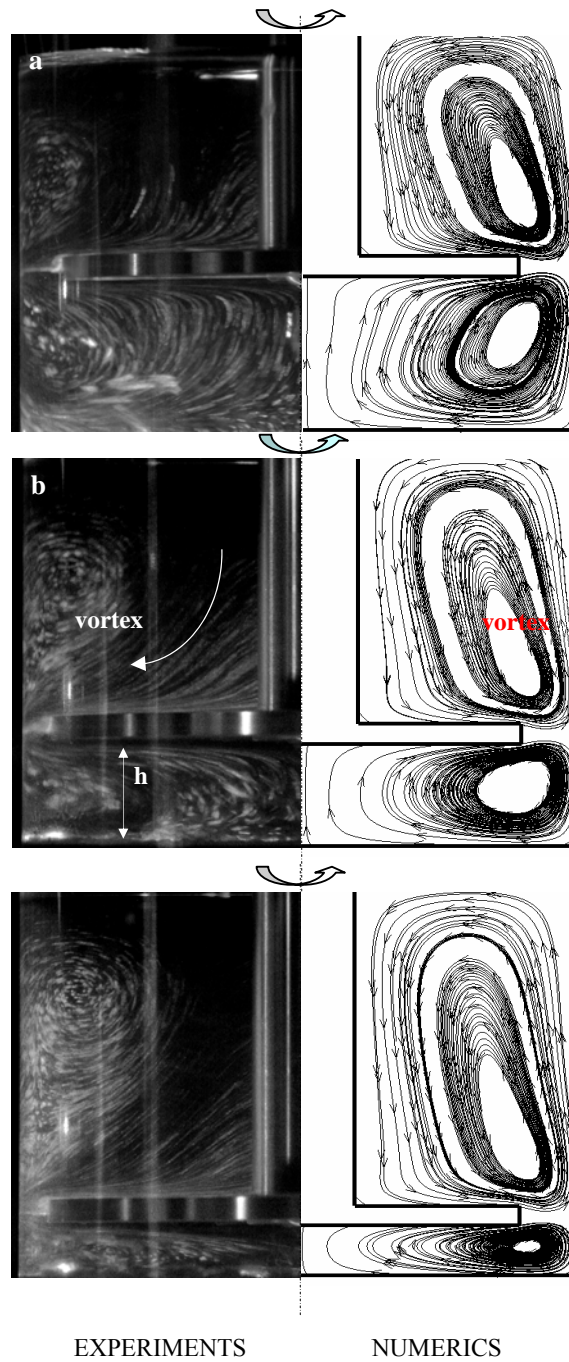


Fig. 3. Experimental and numerical axial-symmetric pathlines distributions in a closed vessel due to rotation of a disc at $Re = 1000$: a) $h = 15$ mm, b) $h = 10$ mm; c) $h = 5$ mm.

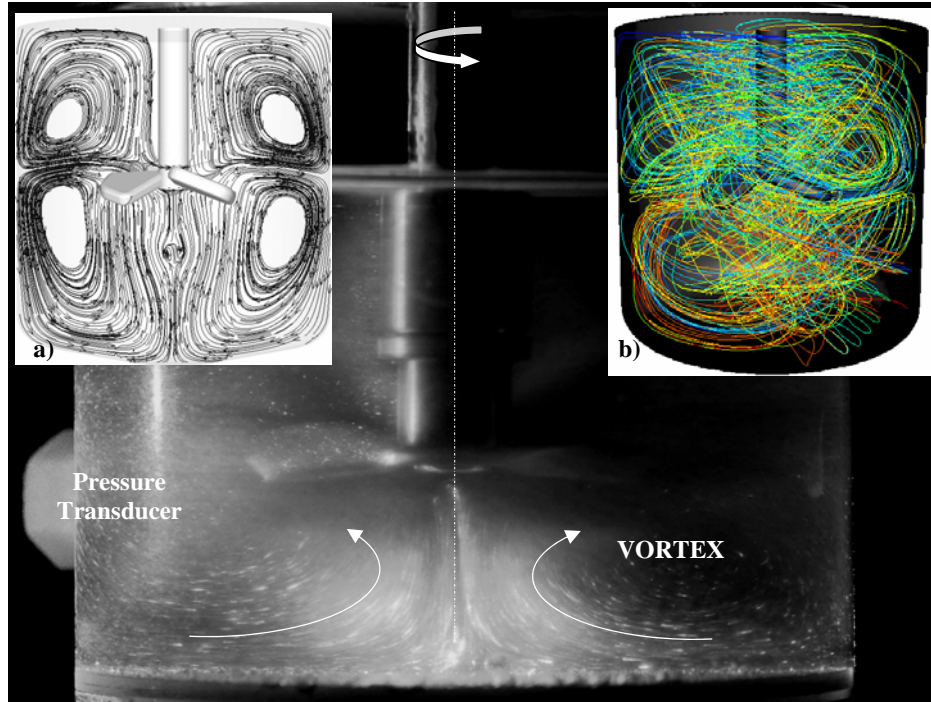


Fig. 4. Experimental flow pattern in the meridian plane generated by the rotation of an impeller in a cylindrical vessel. Numerical streamlines distributions in the meridian plane (secondary flows), axial-symmetric simulations at $Re = 1000$ (a); numerical 3D trajectories at $Re = 1000$ (b).

The outcome of this is an instability mode called wavy Taylor vortex flow (Fig. 5.b) which is characterized by an azimuthal amplitude. The Taylor vortex flow regime was reached using the 2D axial symmetric (Fig. 5.c) and 3D geometries (Fig. 5.d), using in both cases the laminar solver. The Taylor vortex flow regime is the limiting stage for the two-dimensional case, since the wavy Taylor vortex flow is not axial-symmetric. Moreover, the wavy Taylor vortex flow is an unsteady flow regime by nature and therefore the unsteady solver combined with the $k-\varepsilon$ turbulence model was used, in order to numerically develop the instabilities and to obtain the wavy mode, [6].

The last investigations are dedicated to the flows in presence of interfaces. One can classify the motions of two immiscible fluids into three categories: (i) flows in close domains which conserve the mass of the fluids, see Fig. 6, (ii) flows in close domains with variable ratio between the masses of the fluids in contact, see Fig. 7 and Fig. 8, and (iii) flows in infinite spaces, see Fig. 9.

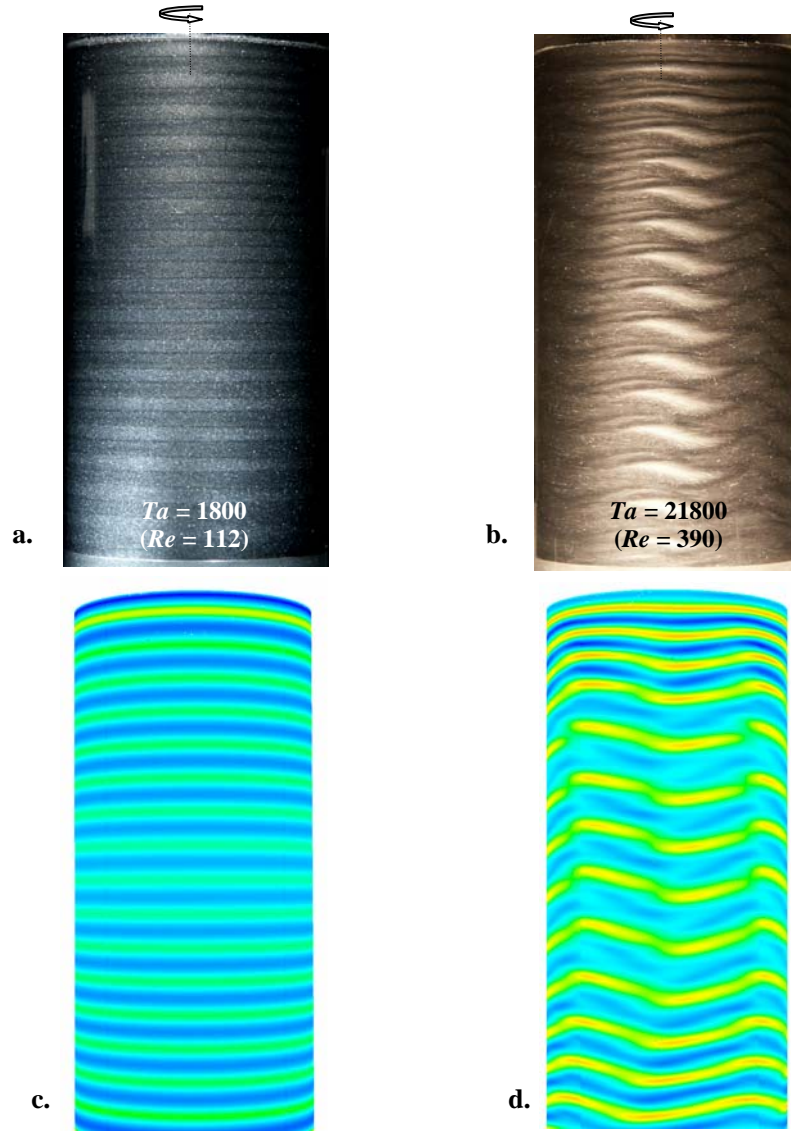


Fig. 5. Experimental visualizations of the flow patterns in Couette-Taylor motion: (a) Taylor vortices, (b) Wavy Taylor vortices. The corresponding numerical simulations are also represented: c) axial-symmetric solution at $Ta = 1800$ (contours of radial velocity in a median cylindrical surface for Taylor vortices flow regime), d) 3D solution at $Ta = 20000$ (contours of radial velocity in a median cylindrical surface of the Wavy Taylor vortex flow regime).

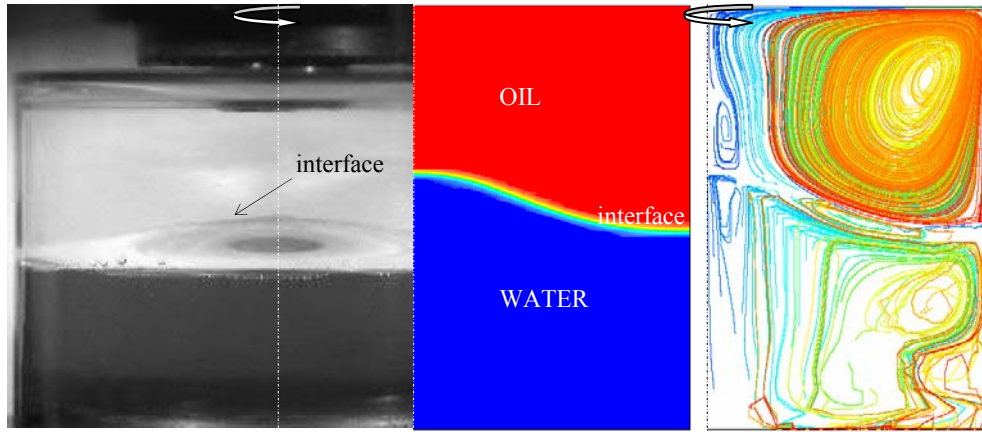


Fig. 6. Interface geometry between water and oil: a) experiment, b) numerical computed interface, c) pathlines distribution (half domain). The motion is generated by rotation of the upper disc, 50 mm diameter, with 75 rot/min. The computations are performed with VOF method, [8].

The first category is represented in Fig. 6, where water and oil are put in motion by a rotating upper disc in a close cylindrical vessel. Influx of one liquid in a close domain filled initially with another liquid and evolution of interface is shown in Fig. 7 and Fig. 8. The geometry of the interface is influenced by the viscosity (elasticity) ratio between the contact liquids and by surface tension property, respectively by the wetting (contact) angle of the fluids with the solid wall of the domain, for details and numerical solutions by VOF, see [7].

The flows of jets and droplets in air represent one of the main application of hydrodynamics of immiscible fluids. In Fig. 9 are shown the detailed pictures of a jet emerged into atmosphere (so called, spinning process). The fluid is in this case a polymer melt (i.e. viscoelastic fluid), which is pulled out through a die by a rolling wheel with different take-up velocities (the exit flow rate melt is constant for all tests). The geometry of the jet (assumed to be axial-symmetric) is given by the dependence of the diameter along the spinning direction, $D = D(Z)$. The respectively function is measured from the pictures and it can be used to compute the extensional viscosity, one main material property of a polymer melt in fibre spinning process (beyond the die swell, the function is monotonic decreasing).

3. Conclusions

The paper presented some of the work performed in the REOROM laboratory in the area of CFD and flow visualizations. CFD is today a very powerful tool to analyze and to characterize complex flow fields, respectively to design novel applications of fluid mechanics in domains of scientific interests.

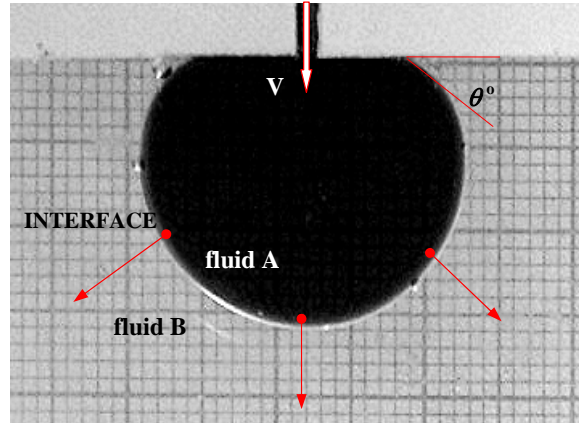


Fig. 7. Instantaneous interface, water in oil, at $V = 0.025$ m/s, with surface tension $\sigma = 0.04$ Pam and $\eta_B/\eta_A = 50$. The interface is here slightly non-symmetric against the position of the inlet. The wall wetting angle θ° is also represented.

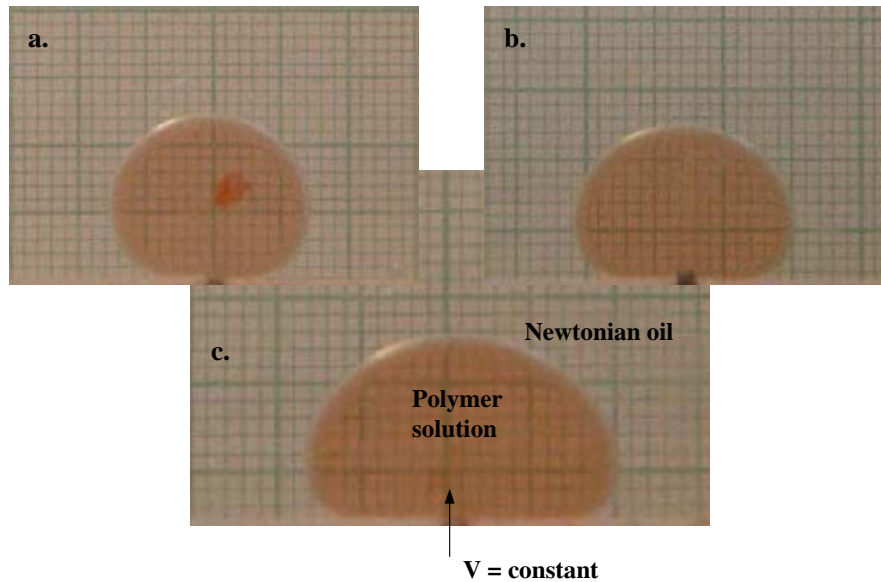


Fig. 8. Experimental shape of the interface between polymer solutions and a Newtonian oil (enter velocity $V = 0.1$ m/s): a) $\eta_0 = 0.66$ Pas, b) $\eta_0 = 13.2$ Pas, c) $\eta_0 = 59.5$ Pas (here η_0 represents the zero shear viscosity of the polymer solution). The flow takes place in a square Hele-Shaw cell with 2 mm gap between the planes and nominal dimension of 150 mm.

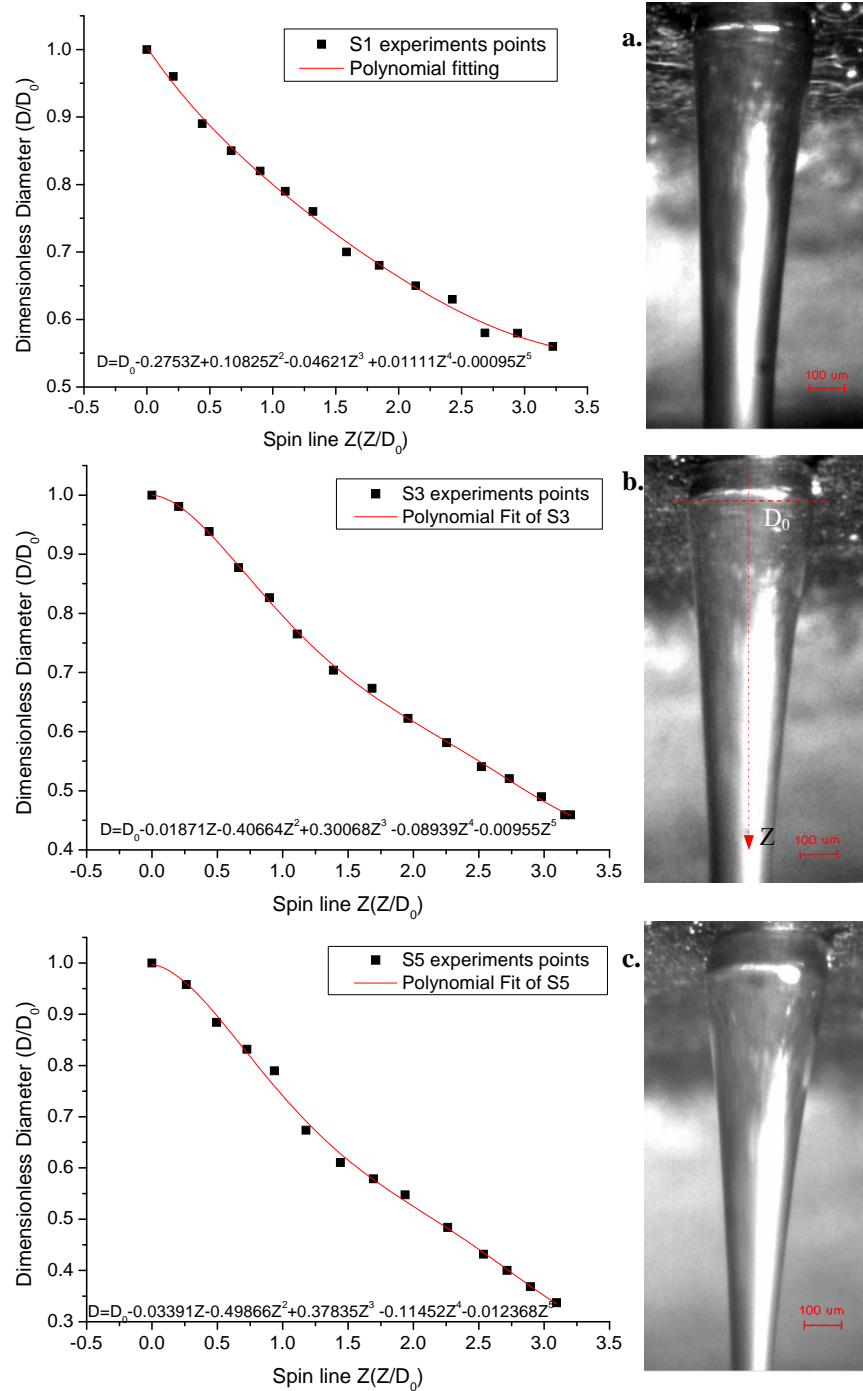


Fig. 9. Dimensionless fibers diameter profiles of a polymer melts and the corresponding pictures; flow rate $64.44 \text{ mm}^3/\text{min}$, die diameter 0.2 mm ; take-up velocity : a) 30 m/min , b) 55 m/min , c) 92 m/min .

Since the real flows take place in 3D geometries, some of them very complex space configurations, it is compulsory to have at hand a validation procedure for the numerical results. Flow visualization techniques are indispensable for any study in fundamental and applied hydrodynamics. Only a good correlation between experiments and numerics can offer a general picture and a correct representation of the flow phenomena and produces results with real potential for practical researches.

The present study had the goal to emphasize some applications of CFD, corroborated with flow visualizations procedures, for modeling various complex flows with relevance in the transport of fluids through channels, mixing of liquids in a chemical reactor, transition of laminar flows to turbulence, flows of immiscible fluids in small gaps and the spinning process of polymer melts.

At the end, one have mention that all investigated flow cases are under study and they will be further developed, adapted and applied in the thesis of the PhD students, who are authors of the paper.

Acknowledgments

The present work was possible through the financial support of the following CNCSIS grants: A-102, TD-14, TD-383 and EU Marie Curie project PolyCerNet, MRTN-CT-019601.

REFERENCES

- [1] *D. Broboana, T. Muntean, C. Balan*, 2005. Mecanica fluidelor cu FLUENT. **vol. 1**, ed. Politehnica Press, Bucuresti (in Romanian).
- [2] *D. Broboana, T. Muntean, C. Balan*, 2007. Experimental and numerical studies of weakly elastic viscous fluids in a Hele-Shaw geometry, Proc. Rom. Academy, Series A, **Vol. 8(3)**, 219–233.
- [3] *D. Broboana, A. Calin, M. Marculescu, C. Balan*, 2006. Vortical structures at the interface, In Workshop on vortex dominated flows – Achievements and open problems, Bucharest, Romania (Ed. by Bernad S. et al.), Sci. Bul. PU Timisoara, Tom 51(65), 63–66.
- [4] *D. Broboana, M. Marculescu, R. Kadar, C. Balan*, 2006. Experimental and numerical simulations of 3D branching flows, with application to the modeling of blood flow in arteries, Timisoara Medical Journal, **Volume 56(4)**, 23–29.
- [5] *H. Schlichting, K. Gersten*, 2000. Boundary Layer Theory, Springer, Berlin Heidelberg.
- [6] *C. Balan, D., Broboana, R., Kadar.,* 2008. Vortex and vorticity, in Vortex Flows and Applications (Ed. by S. Bernad et al.), Eurostampa, Timisoara (in print).
- [7] *D. Broboana, A. Calin, C. Balan*, 2006. Investigation of vortices existence at the interface between immiscible fluids, In CMFF'06, Budapest, Hungary (Ed. by Lajos T. and Vad J.), 806–813.
- [8] *****, 2001. Tutorial Guide, FLUENT Inc.