EUROTHERM 69
Heat and Mass Transfer
In Solid-Liquid Phase Change Processes
June 25-27, 2003, Bistra castle, Ljubljana, Slovenia
Eds.: B. Šarler, D. Gobin,

Experimental model of mould filling flow

Tomasz Michałek* and Tomasz A. Kowalewski

IPPT Polish Academy of Sciences, IPPT PAN, Department of Mechanics and Physics of Fluids
Swietokrzyska 21, PL 00-049 Warszawa, Poland
e-mail: tkowale@ippt.gov.pl

Key words: natural and forced convection, free surface flow, mould filling

Abstract
The aim of our analysis is to provide a simple experimental model simulating the main flow characteristics accompanying casting processes. Hence, a hot liquid is provided under high pressure into an inclined box. The liquid propagates inside the box between two cold isothermal walls, passing obstacles simulating internal complexity of a mould. The main features of the experiment like flow acceleration and deceleration on the obstacle, a free surface flow and sudden increase of the fluid viscosity as it cools down, are typical for a solidification of melt in a mould. Opposite to a real casting, this experimental configuration allows for full control of the experimental conditions and the full field measurements of the temperature and velocity fields. Collection of the quantitative transient data of the flow should permit to verify and validate numerical models used for typical casting problems. The main aim of the investigations is to create an experimental model for validating numerical simulations of mould-filling problems.

1 Introduction

The analysis of free surface flow in thin cavities has become an important issue in the last fifteen years. One of the main reasons is its importance in the modelling and simulation of injection mould filling process. Most industrial injection moulded components are composed of thin sections, often with complex internal geometry. Numerical simulations of the process exhibit several shortcomings and inefficiencies. This is mainly due to difficulties in developing efficient interface tracking algorithm, including full Navier-Stokes flow description coupled with heat and mass transfer [1]. A further difficulty arises when the free surface encounters a solid boundary, either in the form of cavity walls or inserts. In addition solidification and non-Newtonian fluid properties complement the level of numerical difficulties. Therefore, it is not surprising that most of the research publish so far in the area of computational modelling of the injection mould filling process does not adequately address the problem of melt front accuracy, non-uniformity of the melt temperature, formation of void bubbles and fibre-like structure in a solidus. When applied to industrial problems, uncertainties and errors of the numerical modelling are difficult to predict *a priori*, but its failure can be very costly. Due to the problem complexity, the only possibility to find out limitations of the model is to compare it with a properly design experiment, where main flow characteristics can be easily and sufficiently accurate measured and evaluated. Most of industrial configurations and fluids used there do not allow for such measurements. Hence, small-scale experiments using *analog* fluids are often the only alternative for more detailed code validation. In the following we purpose a simple experimental configuration, which allows for direct observation of a free surface flow interacting with internal geometry of the mould model. A transparent model and specially designated experimental technique permits us to evaluate both velocity and temperature fields during the whole filling period. These details are compared with numerical simulations performed using the commercial CFD code Fluent6.1[2]

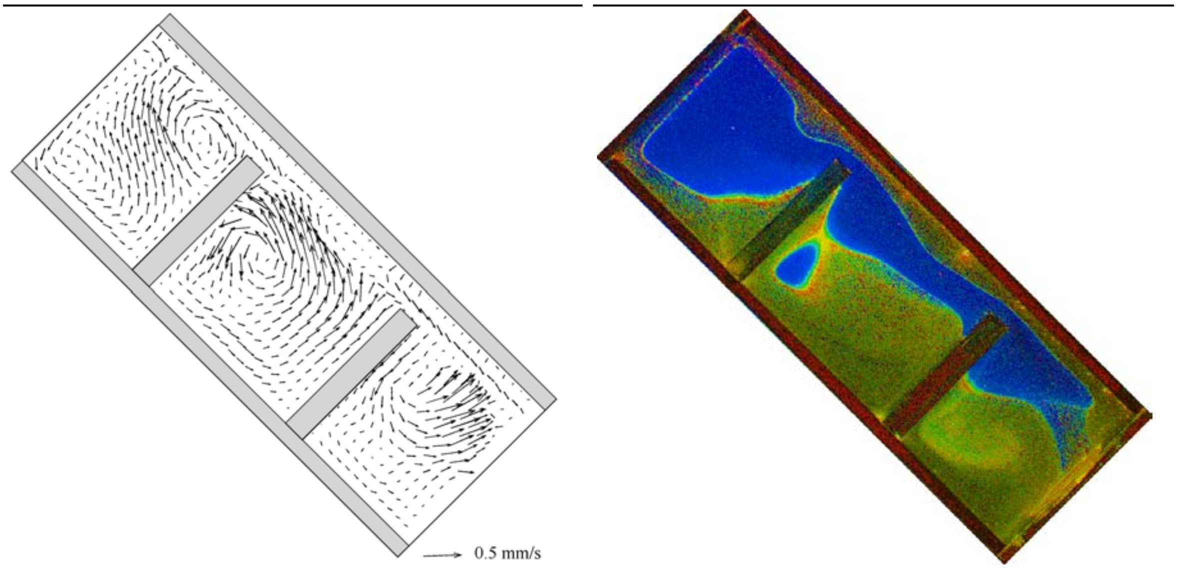
2 Problem Formulation

We consider forced convection in a rectangular, inclined box filled with a viscous liquid. The cavity has square cross-section 38mm x 38mm and is 113mm high. The two side-walls are made from 7.5mm tick Plexiglas. The other two isothermal side-walls are made of copper. They are kept at low temperature Tc =10^{o}C. Two Plexiglas plates located inside cavity are simulating shape complexity of a mould. They form three flow cavities connected only by 5mm high slits between the upper rim of the plates and the cold wall above. The cavity inclination angle α is equal 45^o. More details of the experimental setup can be found by Kowalewski et al. [3] and Sobiecki [4].

The hot fluid of initial temperature T_h =50^{o}C is forced to the cavity at a constant flow rate equal 0.82cm^3/s, through a 13mm circular opening made in the bottom wall. Both forced convection and residual natural convection within the cavity are responsible for the heat transfer through the cold side-walls. The Rayleigh and Prandtl numbers calculated for the experimental parameters are $6\cdot10^6$ and 3720, respectively. The Rayleigh number is based on the maximum temperature difference (40^{o}C), however natural convection starts to play an important role after the inflow is stopped and the temperature differences inside cavity are almost order of magnitude smaller. The Reynolds number for the inflow velocity is very low and at the inlet equals 0.2 only. The non-dimensional numbers are based on the fluid properties defined for the mean temperature (30^{o}C).

Pure 99.5% glycerol is used as the working fluid for its well known physical properties and very strong variation of viscosity with temperature. For the temperature range used the fluid viscosity changes its value almost twenty times, considerably altering a flow pattern at the cold walls. This property of the filling fluid resembles situations typical for solidification of polymeric materials. Also, to some extend, our configuration simulates properties of single fluid phase change models used in numerical simulations, where sudden increase of the fluid viscosity is applied to describe transition to solidus.

Our main interest is directed to collecting quantitative information about the fluid free surface position as well as about velocity and temperature fields within a domain of a mid vertical plane of the cavity. For this purpose the flow images of the centre vertical cross-section have been collected periodically every 3s or 12s, approximately during fifteen minutes from the onset of flow. At each time step series of 3 to 10 RGB images are taken at short time interval. Later on they are used for the flow velocity evaluation (inter images cross-correlation) and for colour evaluation procedures. Particle Image Velocimetry (PIV) and Particle Image Thermometry (PIT) technique is used to obtain 2-D flow temperature and velocity fields[5].



Velocity field (PIV) Temperature field (PIT)

Figure 1. Velocity field (left) and temperature distribution (right) visualized for the cavity shortly after filling process is finished (t=225s). Two cold isothermal walls (upper and lower) are responsible for sudden cooling of the fluid. It changes colour of the liquid crystals seeding from blue (hot) to red (cold regions).

Figure 1 demonstrates an example of the experimental data obtained using this technique for the cavity. The fluid was pumped through the opening in the lower wall. The configuration observed in the figure corresponds to time 225s, shortly after the filling process was stopped. The whole volume of the cavity is filled and the residual velocity and temperature fields are observed. The two-dimensional velocity field is evaluated from a pair of subsequent images, analysing displacement of thermochromic tracers by cross-correlation technique. Three main recirculation zones generated by fluid inertia are well visible. Colour of light scattered by thermochromic tracers indicates local liquid temperature. It changes from blue for higher temperatures, through green and yellow to red for lower temperatures. This happens in a narrow colour play band only, which in our case covers temperature range from 37^{o}

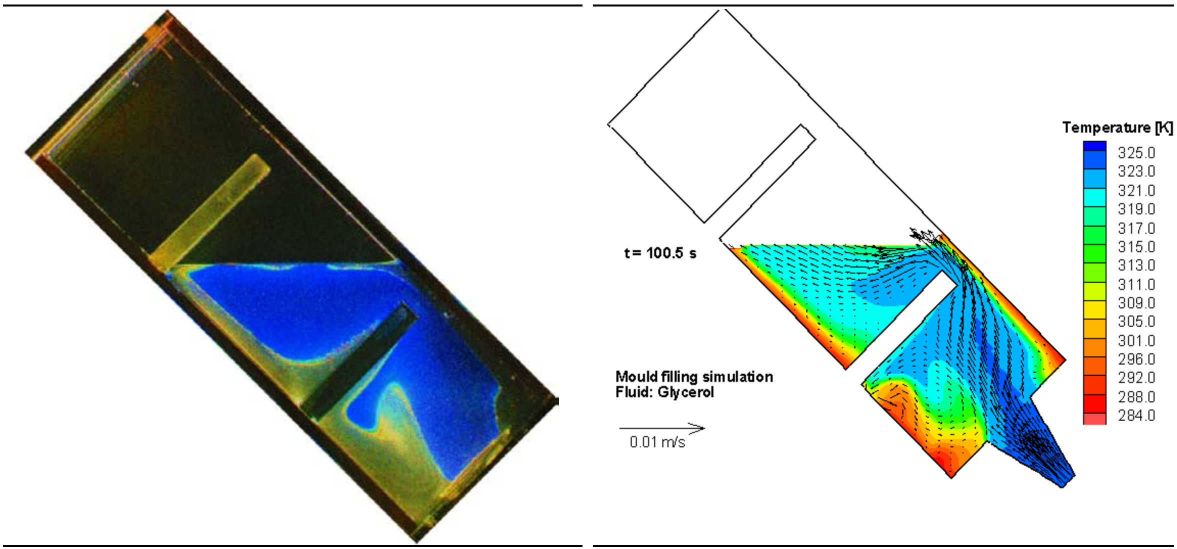
to about 40°C. Regions of nearly uniform blue colour indicate temperatures above this band. Regions of liquid temperatures below the colour play band are visible as more or less colourless, in Fig. 1 little greenish. Accurate temperature measurements are possible within colour play band only. But also indication of general structure of the thermal field is useful for comparing it with adequate numerical results.

The flow of the liquid is simulated by the commercial code Fluent 6.1 [2], the free surface flow behaviour is analysed by an original code using two fluid VOF approach [6] with a laminar flow model coupled with a heat transfer. Two dimensional transient solution is obtained for structured grid with 38x114 nodes. A constant flow rate dynamic boundary condition is imposed for the lower opening of the cavity. For the closing top wall and the bottom wall adiabatic thermal boundary conditions are set. The upper and lower walls are isothermal and set to a constant temperature 10°C, the fluid temperature at the inlet equals 50°C. A constant surface tension for glycerol-air interface and 90° wetting angle are set. By implementing thermal boundary conditions at the walls and variable (temperature dependent) fluid viscosity [4], calculations are performed to simulate the filling of the three sections of the cavity and simultaneous cooling of the fluid along two sidewalls. After the filling process is finished, the zero net flow condition is applied for the inlet and problem of natural convection is solved to describe final cooling of the mould. For the side walls isothermal thermal boundary conditions are assumed. Despite high viscosity of the fluid and relatively simple geometry, simulation performed even for two-dimensional problem appeared to be very costly. To avoid instabilities numerical times steps as small as 10^{-5}s are necessary to initiate the filling process. It is mainly due to inadequate use of the uniform computational mesh, when the cavity is mainly filled only with air. As a result the numerical simulation of the filling process, i.e. about 195s physical time, took almost one month CPU time for 1.7GHz Pentium 4 processor. It seems that application of adaptive mesh, with automatic re-meshing in the regions of interest should largely improve this major shortcoming of the code.

3 Selected results

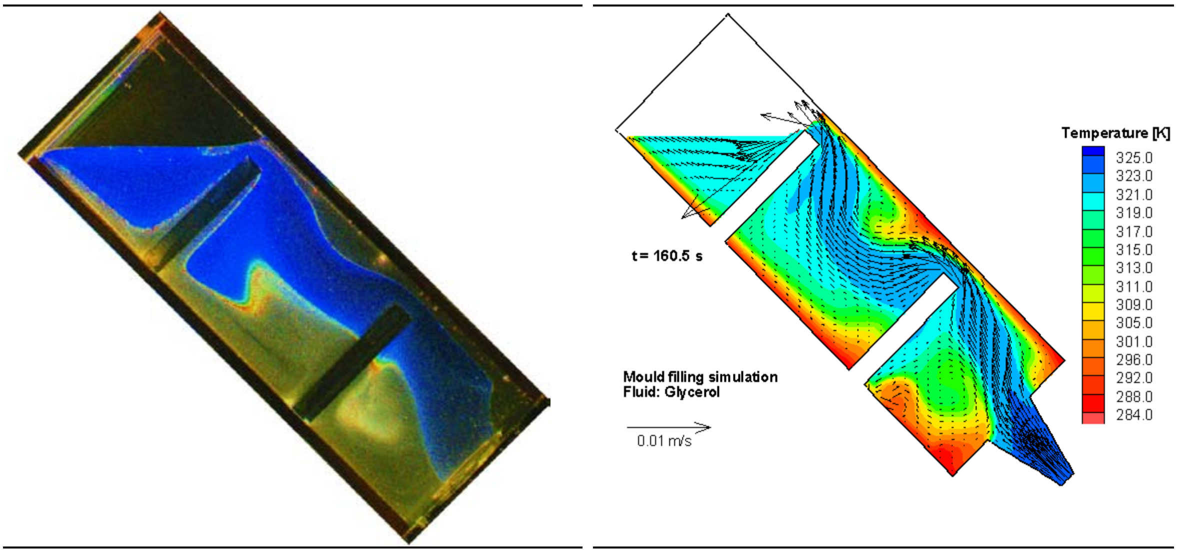
The experiment starts by opening the valve closing connection of the cavity with the pressurized fluid. Initially the hot fluid filling the cavity slips down from the inlet along the bottom wall and cools down as it spreads along the lower isothermal wall. It initiates a clockwise recirculating zone, which finally fills almost the whole part of the first chamber. After the first partition is filled the fluid passes the narrow gap between partitions. Sudden ejection of the fluid through this opening and its penetration down along the partition wall triggers again the recirculation flow, similar to the main vortex observed for the first partition, but rotating in the reverse direction. Due to this recirculation, the flow is well mixed and the temperature distribution at the lower parts of the partitions is quite uniform, except wall regions. When the second partitions fills up, the secondary, smaller recirculation develops along the upper wall. It mainly consists of hot fluid, which is soon ejected through the gap opening to the last partition.

There is only a little amount of fresh, hot fluid transported to the recirculation zone at the lower isothermal wall. Fluid trapped there, cools down relatively fast. Development of highly viscous, cold boundary layer observed in the experiment along isothermal walls is well reproduced by numerical simulation (Fig. 2). The numerical simulation appears to cope reasonable well with a difficult problem of the gap passage. Distribution of temperature and velocity fields are very similar to the experimental one. It is worth noting, especially for the experimental temperature field, development of a hot "tongue" in the first partition, penetrating relatively deep in the cold regions. Such hot intrusions, appearing due to the strong primary recirculations, are often isolated by cold fluid and trapped in *a* *dead flow* region, where fluid cooling is mostly by conduction. Presence of these "hot spots" in a material with high shrinkage coefficient, like metals, leads to creation of void cavities left in the solidifying material.



Experimental temperature field (PIT) CFD temperature and velocity field

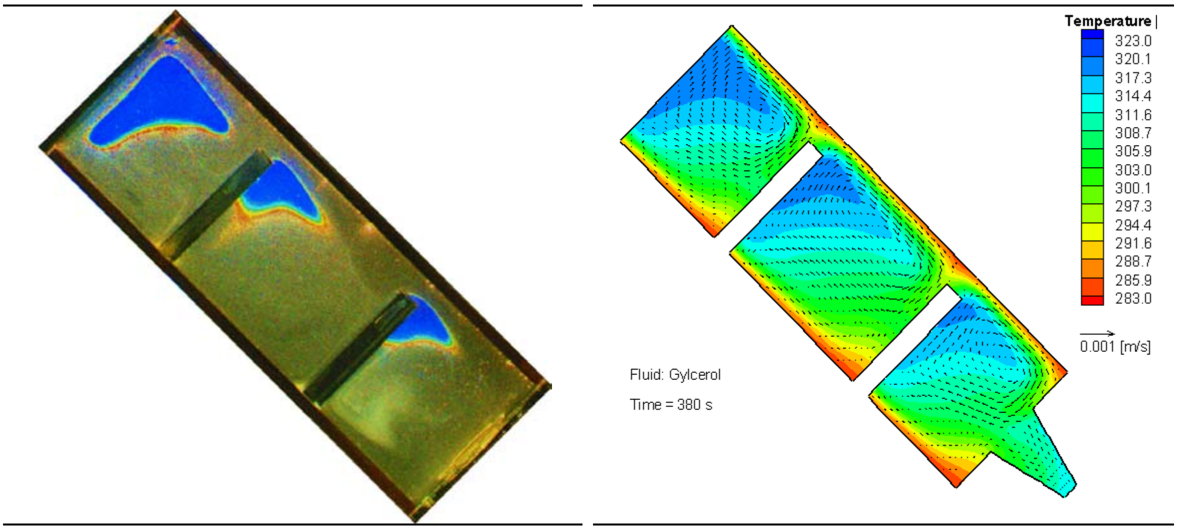
Figure 2. Experimental temperature field (left) and numerical temperature and velocity distribution (right) during filling process after first partition is passed (t=100s).



Experimental temperature field (PIT) CFD temperature and velocity field

Figure 3. Experimental temperature field (left) and numerical temperature and velocity distribution (right) during filling process after second partition is passed (t=160s).

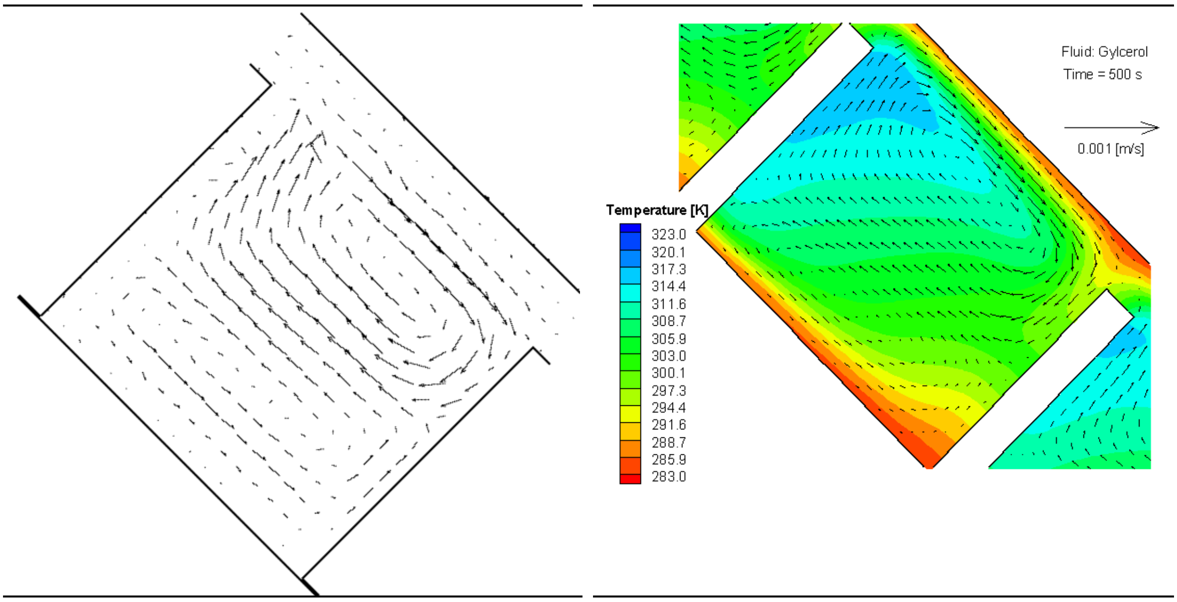
After passing the second gab the temperature field in the second partition attains characteristic "snake like" distribution (Fig. 3). A hot liquid from the inlet meanders inside cavity deflected from the cold upper wall by a heavier cold boundary layer. It seems that due to the strongly increased fluid viscosity, kind of a separation bubble develops at the upper cold boundary in the middle partition. It limits diffusion of heat from the less viscous hot jet below, increasing cooling effect of the wall. It again causes further temperature drop and increase of the fluid viscosity, a self amplified mechanism. Along the upper (second) separation wall a hot "tongue" starts to develop, being later responsible for the "hot spot" well identified when filling of the cavity stops (comp. Figure 1). In the experimental temperature field some deviation in the size of this hot "tongue" are visible. Comparisons of the experimental and numerical velocity fields for the second partition gives some hints for the sources of differences. It appears that a secondary recirculation zone which develops below the "hot snake" is much stronger in the experiment than that in numerical simulation. Such difference remains during whole filling and cooling process, as it can be well seen in Figure 5.



Experimental temperature field (PIT) CFD temperature and velocity field

Figure 4. Experimental (left) and numerical temperature distribution (right) in the cavity during cooling process (t=380s).

The filling time of the cavity equals 195s. After the whole cavity is filled the flow is immediately interrupted. Shortly after then, large clockwise recirculation zones fill all three partitions and cooling process starts. Highly viscous cold fluid does not mix well with the "hot spots" separated form the hot jet during the filling process. They are pushed by buoyancy towards lower surfaces of the partition walls. Figure 4 shows observed and simulated distribution of temperature for the cooling process for time step 380s, almost 200s after inflow is stopped. Three well developed "hot spots" developed in the upper corners of each partition are well visible both in the experimental as well as in the numerical temperature fields. However, the velocity fields show differences, similar to the previously discussed. The lower recirculation region is much less developed in the numerical model, when compared with almost symmetrical double recirculation pattern observed in the experiment (comp. Fig. 5).



Experimental temperature field (PIT) CFD temperature and velocity field

Figure 5. Experimental velocity filed (left) and numerical temperature and velocity distribution (right) during cooling process (t=500s) in the middle partition.

4 Conclusions

The experimental data, collected for glycerol filling a small cavity, are used to validate performance of the VOF method implemented in Fluent CFD code. It appears that the simulation correctly reproduced main features of the free surface behaviour, however there are discrepancies in the flow field pattern. The appearance of the "hot spots" in the simulated temperature fields is less distinct than in the experimental data. Also compared with the experimental data velocity field developed during cooling process seems to be less symmetrical in the numerical model. Detailed analysis of the measured temperature and velocity fields is used to verify sources of the observed discrepancies.

Acknowledgements

The Polish Scientific Committee (KBN Grant No. 8T09A00820) partly supported this work.

References

[1] Rider, W.J., Kothe, D.B., *Reconstructing volume tracking*, J. Comp. Physics, 141 (1998), pp. 112-152.

[2] Fluent 6 Users Guide, Fluent Inc. Lebanon, NH (2002)

[3] Kowalewski, T.A., Cybulski A., Sobiecki T., *Experimental model for casting problems,* in C.A. Brebbia, ed., Computational Methods and Experimental Measurements, WIT Press, Southampton (2001), pp. 179-188.

[4] Sobiecki T., *Modelling of the mould filling process (in polish)*, Diploma (praca magisterska) at Politechnika Warszawska, Plock (2000).

[5] Kowalewski T.A., *Particle Image Velocimetry and Thermometry in two phase problems*, in edts. S. Siedeman & A Landesberg, Visualization and Imaging in Transport Phenomena, Annals of New York Academy of Sciences, vol.972, NY (2002), pp.213-222.

[6] Hirt, C.W., Nichols, B.D., *Volume of fluid (VOF) method for the dynamics of free boundries,* J. Comp. Physics, 39 (1981), pp. 201-225.