HEAT 2002
Baranów Sandomierski, Poland, June 24 – 27, 2002
THE THIRD INTERNATIONAL CONFERENCE ON TRANSPORT PHENOMENA IN MULTIPHASE SYSTEMS

EXPERIMENTAL AND NUMERICAL SIMULATION OF MOULD FILLING PROCESS

A. CYBULSKI, T. MICHAŁEK, T.A. KOWALEWSKI, M. KOWALCZYK[1], M. SOKOLNICKI[2]

ABSTRACT

Experimental model of mould filling process was investigated using water filling a rectangular channel with two cooling walls. Three commercial codes are used to simulate observed flow in purpose to verify their ability to model free surface flow and freezing process.

Keywords: Free surface flow, casting, natural convection, freezing, validation.

INTRODUCTION

The aim of our analysis is to provide a simple experimental model simulating the main flow characteristics accompanying casting processes. The hot fluid is provided under pressure into an inclined box. Fluid propagates inside the box between two cold isothermal walls, eventually passing obstacles simulating internal complexity of a mould. Water is used as a working fluid. Hence, when temperature of the isothermal walls is set below the freezing point, solidification is observed. The main features of the experiment like a free surface flow and freezing are typical for a solidification of melt in a mould. Opposite to a real casting process, 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 strongly dynamic effects accompanying the flow create a challenging problem for numerical simulations of free surface flows. Several numerical codes use Volume of Fluid (VOF) method for simulating free surface flow and a fixed mesh enthalpy

[1] A. Cybulski, T. Michałek, T.A. Kowalewski, M. Kowalczyk – IPPT PAN, Polish Academy of Sciences, Swietokrzyska 21, 00-049 Warszawa, Poland.
[2] M. Sokolnicki – Instytute of Implementation and Technology Ltd, Wyzwolenia 70, 27-200 Starachowice, Poland.

method for solidification. Both these methods introduce severe simplifications of physics of the phenomenon. Hence, it is useful to validate them against experimental data, even if we operate at conditions different from the typical casting process. In the following we describe our preliminary attempts to simulate the experimental results using three commercial codes FLUENT® (Flunet Inc, USA), Procast® (Calcom SA, Switzerland), and Vulcan (Quantech, University of Barcelona). The two last codes are finite-element solvers explicitly designed to simulate casting problems.

1. PROBLEM DESCRIPTION

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 113 mm high. The two sidewalls are made from 7.5 mm tick Plexiglas. The other two isothermal sidewalls are made of copper [1]. They were kept at low temperature Tc. The cavity inclination angle α varied from 12° – 78°. The hot fluid of initial temperature T_h was forced to the cavity through a 13 mm 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 sidewalls.

As working fluids, water was used to simulate high Reynolds number free surface flow interacting with the mould walls, as well as the phase change effects. The temperature range applied in the freezing experiments was $T_{c=}$ -15°C for the cold isothermals walls, and T_h = 3°C as an initial fluid temperature.

The temperature of isothermal walls and the initial fluid temperature were maintained at a constant value by two thermostats. The experiment started by opening abruptly the inlet valves at the bottom wall for the pressurised fluid. The flow rate through the inlet was set in the range form $1\cdot10^{-6}\text{m}^3/\text{s}$ to $5\cdot10^{-6}\text{m}^3/\text{s}$.

The experimental set-up used to acquire temperature and velocity fields consisted of a 3CCD colour camera and a 32-bit PCI bus frame grabber. The 24-bit colour images of 768x256 pixels were acquired using a Pentium III computer. The flow field was illuminated with a 2mm thin sheet of white light from the halogen lamp, and observed in the perpendicular direction. Both velocity and temperature fields were monitored using unencapsulated Thermochromic Liquid Crystal (TLC) tracers. Digital evaluation of colour of the tracer images collected for the centre flow cross-section was used for simultaneous and fully automatic measurements of temperature and velocity 2-D flow fields (PIV&T – Particle Image Velocimetry and Thermometry [2]).

The dimensionless parameters defining the problem are the Rayleigh and Prandtl numbers: $Ra = g\beta(T_h - T_c)L^3/\kappa\nu$, $Pr = \nu/\kappa$. In the above definitions g, L, κ, β,ν and ρ denote respectively the gravitational acceleration, cavity height, thermal diffusivity, coefficient of thermal expansion, kinematic viscosity and density of the liquid. To account for the inertial effects, the Reynolds number, $Re = Q/D\nu$, is defined using the imposed flow rate Q and the cavity width D. In the experiments

Ra varied from $10^7 - 10^8$, $Pr = 8$, $Re = 100 - 500$. Our main interest was directed to the collection of quantitative information about the velocity and temperature fields, as well as on the shape and position for fluid/gas and fluid/solid interface.

Numerical simulations of the problem were mainly performed using a finite volume code Fluent, with a laminar viscous flow model and VOF method for free surface flow simulation. For water as a working fluid, the basic physical properties: viscosity, thermal conductivity, and specific heat were assumed to be constant. An anomalous thermal variation of the water density implemented in the numerical code was described by the fourth order polynomial, given elsewhere [3]. In the simulations performed with the two casting codes constant thermal properties of water were assumed (Boussinesq approximation). Plexiglas walls were assumed to be adiabatic and the fluid temperature at the inlet to be constant. The thermostated sidewalls were assumed to be "truly isothermal". The last assumption appeared to be far from the reality, and a part of the simulations was repeated applying in the code measured temperatures for the both metal walls.

2. SELECTED RESULTS

In the following we describe an experiment with the cavity inclined at 78.6°, filled with water at constant flow rate $4.6 \cdot 10^{-6} m^3/s$. The experiment starts after opening the inlet valve at the bottom wall. Due to the inclination, fluid film slips down through the opening gap. It takes about 5s until the surface smoothes out and the regular propagation of the flat interface recovers. The effects of fluid inertia evidently deform the surface, exciting surface perturbations at the beginning of the filling process. Fluid falling down from the inlet along the bottom wall fills the lower part of the cavity and cools at the lower isothermal wall. It generates the main clockwise recirculating zone in the lower part of the cavity. After about 20s almost half of the cavity is filled. Several counter-rotating recirculation zones develop with time, intensifying heat transfer from the lower isothermal wall. One of them is well visible in Figure 1a. Close to the bottom wall the temperature distribution (Fig. 1b) exhibits thermal stratification with isotherms being almost parallel to the wall. In addition it appears that patch of cold fluid propagates along the fluid surface.

The numerical simulations performed with Fluent relatively well describe initial disturbances of the free surface observed during first seconds of the filling process. It includes drop-wise sliding of the first masses of fluid through the gap. The flow structure in the cavity (Fig. 2a) indicates existence of several counter-rotating structures, similar to the one observed in Figure 1a. However, in the numerical model the inflow through the orifice seems to generate strong perturbation injecting warm fluid into cavity, effect not detected in the experimental study. Such perturbation intensifies mixing process and cooling of the bulk fluid. After about 20s, very thin layer of ice created at the lower wall is present in the simulation.

Results of the second numerical simulation performed with Procast (Fig. 2b) describe rather uniform temperature in the whole cavity, except strong gradients close to the cooling wall. We may also find that surface disturbances are not smoothed out. Similar results are obtained using the other casting code Vulcan.

a) b)

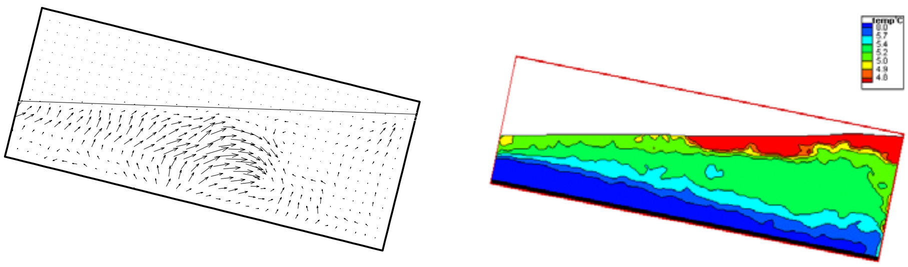

Fig. 1. Water filling cavity and freezing at the side walls after 20s: a) PIV evaluated velocity field; b) PIT evaluated isotherms. Inclination α = 78.6°, T_c = -15°C, T_h = 3°C, Q = $4.6 \cdot 10^{-6}$ m^3/s

a) b)

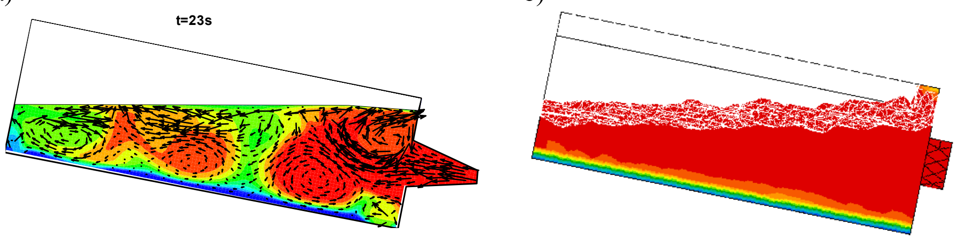

Fig. 2. Numerical simulations of the case from Figure 1: a) temperature and velocity fields from Fluent, b) fluid surface and temperature distribution from Procast

The solidification process may start immediately after first layer of water gets in contact with the cold lower surface. However, in the experiment there is at least 20 s delay before water commences to freeze. This delay is due to the finite time for building first nucleation sites and it depends on water supercooling, surface structure and purity of water. Usually numerical codes do not take into account such effects. Hence, the casting programs exhibit solidification from the first seconds of the filling process. Results generated with Fluent are closer to the experimental data, first patches of ice can be found after about 20s. It is probably due to the strong initial mixing process, which appears in these simulations and prohibits uniform thermal stratification at the bottom wall.

After completing the filling process the flow is stopped and freezing from both surfaces dominates. The measured and calculated flow velocities are very small.

Apparently, there is only residual convective motion, which has little effect on the phase change process. Initially the growth rate of ice at the lower wall is faster than at the upper one. This surface was first covered by water, and initial layer of ice exists there before the flow is stopped. But after about 4 minutes this difference vanishes completely and the uniform growth of the ice layer is observed for both walls.

All numerical simulations predict uniform growth of the ice layer of the same thickness on both walls. The initial delay in covering the upper wall has apparently negligible effect for further freezing. Initially, serious discrepancies between simulations and experiment were noted for the ice growth rate (Fig. 3). It was found however, that the thermostated metal walls did not maintain their initial temperature (-15°C), and corrected values measured during the experiment had to be used instead. Such corrected calculations, performed only with Fluent, show significant improvement of the predicted growth rate. The simulated growth rate of the ice layer at the lower wall appears to be close to the experimental data (Fig. 3). However, the growth rates predicted for the upper and lower walls are similar after the filling process stops. In the experiment it is necessary to wait about 180s for equal thickness of the ice at the both walls. This discrepancy is probably due to the initial warming of fluid by the mixing process and release of the latent heat at the lower wall. Observed increase of the temperature close to the freezing surface may indicate such effect. Such effect seems to be absent in the present simulations. To improve the modelling, additional investigations will be undertaken to take into account heat transfer during the cavity filling time, and to solve the conjugate heat transfer problem including all side walls of the cavity.

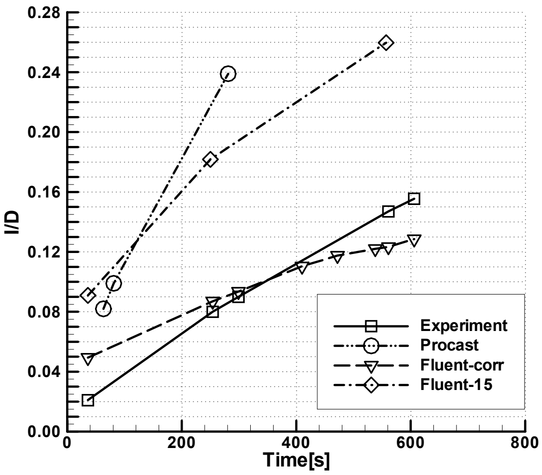

Fig. 3. Non-dimensional thickness of ice layer at the lower wall, cavity inclination angle $\alpha = 78.6°$, initial fluid temperature $T_h = 3°C$. Solid line – experimental data, dashed lines – numerical prediction using: Fluent and Procast for idealized boundary conditions (i.e $T_{cl} = -15°C$), and Fluent with corrected thermal boundary conditions (measured values for lower wall temperature $T_{cl} = -7°C$ and upper wall temperature $T_{cu} = -6°C$)

3. CONCLUSIONS

The simple experimental model of the mould was used to study a free surface flow and solidification. It allowed for the collection of quantitative data on the flow pattern, the velocity and temperature fields, and the interface shape. Preliminary numerical study of the experimental cases shows only qualitative agreement. Several details differ, like formation of the recirculating zones, and the growth rate of the solid phase, sometimes quite seriously. It appears that numerical modelling needs additional improvements. Its verification is eased with the help of the available experimental data.

REFERENCES

1. Kowalewski T.A., Cybulski A., Sobiecki T., Experimental model for casting problems, in Computational Methods and Experimental Measurements, pp. 179-188, WIT Press, Southampton 2001.
2. Kowalewski T.A., Cybulski A., Rebow M., Particle Image Velocimetry and Thermometry in Freezing Water, Proc. 8th Int. Symposium on Flow Visualization, CD ROM Proceedings ISBN 0953399109, pp. 24.1-24.8, Edinburgh, 1998.
3. Kowalewski T.A., Rebow M., Freezing of water in the differentially heated cubic cavity, Int. J. of Comp. Fluid Dyn., Vol. 11, pp.193-210, 1999.

Acknowledgements

This work was supported by Polish Scientific Committee (KBN Grant No. 8T09A00820). Parts of the computations were performed at the computer centre of the Warsaw University of Technology (COI).