Söller C., Hiller W.J., Kowalewski T.A. & Leonardi E.

Experimental and numerical investigation of convection in lid cooled cavities — effects of non-ideal thermal boundary conditions on three-dimensional flow

The flow structures observed in a lid cooled cavity were investigated with the aim to understand and explain discrepancies between measured and calculated flow patterns. It was found that the thermal boundary conditions of non-isothermal walls have triggering effect on the flow configuration. It indicates that simplifications or idealizations usually applied in numerical models may lead to substantial differences of the flow pattern, even if the global parameters (like Nusselt number) apparently are in a good agreement. Solving the coupled solid-fluid heat conduction problem together with the 3-D Navier-Stokes equations improved modelling of the flow pattern.

1. Formulation of the problem

The problem of melt-flow in a lid-cooled cavity has practical application in a number of manufacturing processes and physical situations. A large scale example is the cooling or freezing of water reservoirs, on a smaller scale such a flow configuration is important in crystal growth technology.

Our aim was to investigate both experimentally and numerically the effects of non-ideal thermal boundary conditions of the "side" walls on the flow structure and thermal fields in natural convection, when the flow is generated due to cooling from above. The problem emerged during previous investigations of ice growth, where difficulties appeared in matching details of experimental and numerical results [1]. The present investigations were performed for a 38mm cubical box filled with an incompressible, viscous, heat conducting liquid (water). Steady, laminar natural convection at moderate Rayleigh numbers ($Ra \approx 10^6$) was considered. The lid of the cavity was practically isothermal (metal), whereas the other five walls made of 2mm thick glass or 8mm thick plexiglas were not truly either isothermal or adiabatic. The box was immersed in an external water bath of a constant temperature. Due to forced convection in the bath it can be assumed that the temperature at the external surfaces of the box was constant. The heat fluxes into the box through the sides and bottom, as well as out through the lid are responsible for the generation of the flow. An attempt was therefore made in our computations to model this "leakage" of heat into the test fluid. The lid temperature and the temperature of the the external fluid were used to define the Rayleigh number for the problem.

2. Experiment

For the flow visualization the seeding with thermochromic liquid crystals was used. It allowed instantaneous measurement of the velocity and temperature fields in two dimensional cross-sections of the cavity. Two digital methods employing Particle Image Velocimetry and Particle Image Thermometry techniques were used for this purpose [2]. A novel tomographic technique using Digital Speckle Holographic Interferometry was used to obtain three dimensional temperature fields [3]. To visualize the flow structures, paths of the individual tracers have been followed using multiexposure.

3. Numerical

A numerical simulation of the problem was performed using a finite difference vorticity-vector potential formulation of the Navier-Stokes and energy equations. A modified version of the 3-D numerical code FRECON3D [4] has been used. It is a relatively robust, false transient solver allowing quick generation of the steady state solutions, ideal for parametric studies like the one described. In the computational modelling several approaches were adopted to simulate as closely as possible the physical experiment. Specifically, the Thermal Boundary Conditions (TBC) for the non-isothermal walls have been either computed using the 1-D or 3-D modelling of the heat flux through/along these walls. In the first 1-D TBC approach an arbitrary temperature, a specified heat flux or a specified heat transfer coefficient on each of the six surfaces of the box was imposed in calculations. The general form of non-dimensional conditions for the temperature θ at the inner boundary was used:

$$a\theta + b\frac{\partial\theta}{\partial n} = c \tag{1}$$

The constants a, b and c were varied to specify the boundary condition existing in the experiment.

In the 3-D TBC case the additional energy equation for the physical walls has been incorporated into the numerical model, and the coupled fluid-solid heat conduction problem was solved.

Solutions were obtained using a $61 \times 61 \times 61$ uniform mesh.

4. Results

The flow visualisations have shown the existence of a complex spiralling structure transporting fluid up along the side walls and down in a central cold "jet" along the cavity axis. For high heat conducting walls (glass) eight symmetric cells were created by the flow. For plexiglass walls additional small recirculation regions appeared separating the main cells. It was also found that disturbances of TBC at the lid-side walls connection could introduce generation of small spiralling structures in the thermal boundary layer directly underneath the lid.

Although the computational results obtained for "idealized" 1-D TBC confirmed the eight fold symmetry of the temperature and flow fields observed experimentally, their orientation was different. Also measured temperature values were almost 30% higher than predicted. In the several computational runs TBC in (1) were step-wise modified so as to give better agreement with measured temperature profiles. It was found that such agreement could be obtained by assuming nearly twice as much heat flux through the side walls as the nominal value calculated from the physical characteristics of the side walls. It indicated that the heat flux along the side walls, neglected in (1), effectively increases the heat transfer from the external medium. The differences between observed and calculated flow patterns remained even after such arbitrary modification of TBC. Consequently calculated streamlines starting in the diagonal symmetry plane spiraled in the opposite direction of that observed in experiments. Especially, for "thick" plexiglas walls TBC could not be properly modelled using a 1-D assumption about the heat flux.

Physically it is possible for a flow pattern with the opposite sequence to develop, that is spiralling inwards in the central plane and outwards on the diagonal plane. Therefore only slight change of the TBC may modify the flow pattern. This was observed by replacing the side walls of low conductivity plexiglas by thin glass walls.

The numerical simulations performed for both cases using the 3-D model for wall heat conductivity confirmed the triggering mechanism of TBC on the observed flow pattern. Inclusion of the side walls in the computational domain and solving the coupled fluid-solid heat conduction problem improved agreement with the observed flow pattern. Also the observed temperature distribution as well as its symmetry were fully recovered in the numerical results. It was only as a result of the use of *both* experimental and numerical methods that the fine structures of the thermal flow were understood fully. This is due to the difficulties in predicting changes of the flow pattern, which are triggered by a minor modification of the thermal conditions at usually idealized "side walls".

Acknowledgements

The authors gratefully acknowledge a generous participation of Graham de Vahl Davis of the University of New South Wales in the present work.
The third author acknowledges support from the Polish Scientific Committee (KBN Grant No. 3P40400107).

5. References

1 Abegg C., de Vahl Davis G., Hiller W.J., Koch St., Kowalewski T.A., Leonardi E., Yeoh G.H.: Experimental and numerical study of three-dimensional natural convection and freezing in water; Proc. of 10th Int. Heat Transfer Conf. **4**, 1-6, Edt. G.F. Hewitt, IChemE 1994.
2 Hiller W.J., Koch St., Kowalewski T.A., Stella F.: Onset of natural convection in a cube; Int. J. Heat Mass Transfer **36** (1993), 3251–3263.
3 Söller C.: Untersuchung der Konvektionsströmung in von oben gekühlten Behältern mittels tomographischer Speckle-Interferometrie und digitaler Bildverarbeitung, Dissertation, University of Göttingen, 1994.
4 Goh L.P., Leonardi E., de Vahl Davis G.: FRECON3D - Users Manual. A program for the numerical solution of mixed convection in a three- dimensional rectangular cavity; Report 1988/FMT/7, UNSW, Sydney 1988.

Address: C. Söller, W.J. Hiller, Max-Planck-Institut für Strömungsforschung, D-37073 Göttingen, Germany
T.A. Kowalewski, Polish Academy of Sciences, IPPT PAN, PL 00-049 Warszawa, Poland
E. Leonardi, University of New South Wales, 2052 Sydney, Australia.