Advanced Computational Methods in Heat Transfer

Heat Transfer V

EDITORS:

A.J. Nowak
Silesian Technical University, Poland

C.A. Brebbia
Wessex Institute of Technology

R. Białecki
Silesian Technical University, Poland

M. Zerroukat
Wessex Institute of Technology, UK

Computational Mechanics Publications
Southampton, UK and Boston, USA

Professor A.J. Nowak
Silesian Technical University
44-101 Gliwice
Konarskiego 22
Poland

Professor R. Białecki
Silesian Technical University
44-101 Gliwice
Konarskiego 22
Poland

Professor C.A. Brebbia
Wessex Institute of Technology
Ashurst Lodge
Ashurst
Southampton SO40 7AA, UK

Dr M. Zerroukat
Wessex Institute of Technology
Ashurst Lodge
Ashurst
Southampton SO40 7AA, UK

Published by

Computational Mechanics Publications
Ashurst Lodge, Ashurst, Southampton SO40 7AA, UK
Tel: 44(0)1703 293223; Fax:44(0)1703 292853
Email: cmp@cmp.co.uk
http://www.cmp.co.uk

For USA, Canada and Mexico

Computational Mechanics Inc
25 Bridge Street, Billerica, MA 01821, USA
Tel: 978 667 5841; Fax: 978 667 7582
Email: cmina@ix.netcom.com

British Library Cataloguing-in-Publication Data

A Catalogue record for this book is available
from the British Library

ISBN: 1 85312 591 1 Computational Mechanics Publications

ISSN: 1462-6063 Computational Mechanics Publications

Library of Congress Catalog Card Number 98-84576

The texts of the various papers in this volume were set individually by the authors or under their supervision. Only minor corrections to the text may have been carried out by the publisher.

Printed and bound in Poland

Free surface natural convection in a differentially heated rectangular cavity

T.A. Kowalewski[*], G. de Vahl Davis[†] and E. Leonardi[†]
[*]*IPPT PAN, Polish Academy of Sciences, PL 00-049 Warsaw.*
Email: Tomasz.Kowalewski@ippt.gov.pl
[†]*University of New South Wales, Sydney, Australia 2052.*
Email: G.deVahlDavis@unsw.edu.au, E.Leonardi@unsw.edu.au

Abstract

An experimental and numerical study has been made of natural convection in an open cavity. It was found that the presence of a free surface increases the hot-cold asymmetry of the bulk flow. At the free surface, a very slow recirculation was detected which appears to be decoupled from the main circulation in the cavity. Using standard free-surface boundary conditions, numerical modelling properly reproduced the bulk flow characteristics, but not the surface flow. A novel kinematic representation of the surface layer enabled the experimentally observed surface flow to be reproduced numerically.

1 Introduction

Free surface flow is attracting increasing attention due to new material processing technologies such as laser welding, glass processing and the growth of high purity crystals. However, relatively little experimental or numerical work has been done for three-dimensional flows. Hence, our aim is to examine the effects of the free surface on natural convection for the popular bench-mark configuration, *i.e.* a differentially heated box. In comparison with flow in a closed cavity, is this drastic change of kinematic boundary conditions crucial for the overall flow structure? To answer this question, experimental results have been gathered for both an open and a closed cavity. The flow has also been modelled by a three-dimensional numerical simulation using a finite difference method. We suggest that a proper interpretation of the observed surface flow needs a modification to the usual surface kinematic boundary conditions.

2 Problem Formulation

We consider convective flow in a cubical box filled with liquid. The origin of a rectangular Cartesian coordinate system is located at the lower, back corner of the cavity, with the y axis directed upwards; the hot and cold walls are at x=0 and x=1, respectively. In the free-surface experiments, the cavity was filled to y=0.9, leaving a gap of air above the surface. The motion is driven by a temperature difference T_h–T_c applied to two opposite side walls of the cavity. The other four walls, made of plexiglas, are theoretically adiabatic but in practice not so. To examine how the free surface affects the flow structure, a parallel set of experiments was performed in which the cavity was closed by an additional plexiglas plate at y=0.9, thus preserving the geometry of the flow domain. The dimensionless characteristic parameters defining the problem are the Rayleigh and Prandtl numbers: $Ra=g\beta(T_h-T_c)L^3/\kappa\nu$, $Pr=\nu/\kappa$. To account for the effects of surface tension at the free surface, two additional non-dimensional numbers characterise the problem, the Bond number $Bo=g\rho L^2/\sigma$ and the Marangoni number $Ma=\gamma(T_h-T_c)L/\kappa\nu\rho$. In the above definitions g, L, κ, β, σ, γ, ν and ρ denote respectively the gravitational acceleration, cavity length, thermal diffusivity, coefficient of thermal expansion, surface tension, surface tension gradient $|\partial\sigma/\partial T|$, kinematic viscosity and density of the liquid.

3 Experimental Apparatus

Selected cross sections of the flow domain were illuminated by a light sheet and observed by a high resolution 3CCD colour camera connected to a three-channel frame grabber. The computer controlled system with step motors allows automatic scanning of the flow domain and collection of a 3D picture of the flow within a short time. For flow visualisation, seeding with thermochromic liquid crystals is used. This allows instantaneous measurement of the velocity and temperature for 2D cross-sections of the cavity, using Particle Image Velocimetry and Particle Image Thermometry (Hiller et al.[1]). To obtain a better view of the flow structure, a series of black and white images was also accumulated. The images were superimposed to produce particle tracks.

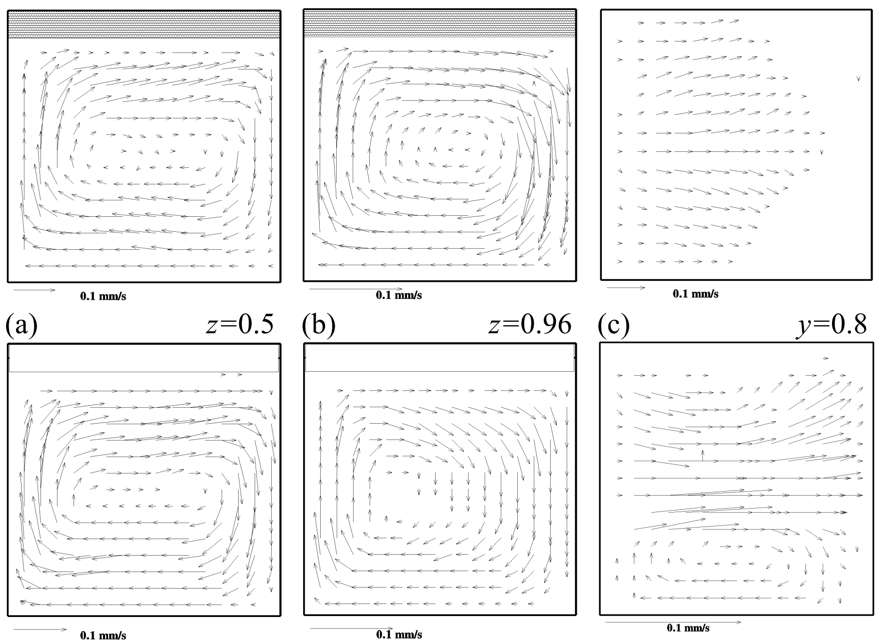
The cavity is a 50 mm (internal) cube. Two opposing vertical walls of aluminium were heated to different temperatures T_h and T_c by temperature-controlled water. The remaining walls are made from 8 mm thick plexiglas. Initially both the isothermal walls and the liquid were at temperature T_c. The flow was started by suddenly opening the inlet valves to the heating passages in the hot wall. The temperature difference applied was 3K or 5K (± 0.05K). The experiments were performed with the cavity filled with several water-glycerol solutions, allowing a variation of properties, especially the fluid viscosity. The liquid was observed to wet the plexiglas walls, forming a small rising meniscus there. Its radius of curvature was below 1 mm. The Rayleigh number corresponding to the experimental conditions covered a range of 22,000–395,000 and the Prandtl number was in the range 752–8,900. The Bond and Marangoni numbers were 492 and 99–2,060 respectively. It follows that $Ma/Ra \sim O(0.005)$ and thus thermocapillary and surface deformation effects are negligibly small compared with the buoyancy effects.

4 Numerical Simulation

A numerical simulation of the problem has been performed using a finite difference model of the Navier-Stokes and energy equations. We assume a flat surface and the absence of thermocapillary flow. A modified version of the 3D code FRECON3D (from UNSW) was used to simulate steady flow. It uses the vorticity/vector potential formulation and, to enhance convergence, incorporates the false transient method (Mallinson & de Vahl Davis[2]). The equations are parabolised in time and their solution is only correct in the steady state. For some simulations the Boussinesq approximation was used, in which the density is assumed constant except in the buoyancy term of the equation of motion. The thermal conductivity, viscosity and specific heat of the fluid were also assumed to be constant. A modified version of FRECON3D was also used to quantify the effects of variable liquid properties (viscosity and thermal conductivity). Time development of the flow pattern was simulated using the true transient vorticity-vector potential code FREEZE3D (also from UNSW). A feature of this code is the use of body-fitted coordinates, here used to test the effects of surface curvature due to the meniscus. Most of the computations were performed using a uniform 31^3 mesh, selected as a compromise between accuracy and speed.

In each code, the implementation of the thermal boundary conditions is sufficiently flexible to allow the imposition of an arbitrary temperature, a specified heat flux or a specified heat transfer coefficient on each surface of the box. The heat flux was estimated using heat transfer theory applied to a thick wall of uniform conductivity next to an external unlimited environment. Standard no-slip conditions were imposed on the rigid walls. The free upper surface ($y=0.9$) was assumed to be flat, corresponding to a fluid with a contact angle of 90º. This assumption is not very restrictive. Numerical tests modelling a 1 mm meniscus at the contact line showed that such a small surface deformation does not affect the flow structure. Hence, in all further simulations the effect of the meniscus was neglected. The kinematic boundary conditions on the free surface are $u_y = \partial u_x/\partial y = \partial u_z/\partial y = 0$.

5 Results



(a) z=0.5 (b) z=0.96 (c) y=0.8

(d) z=0.5 (e) z=0.9 (f) y=0.9

Figure 1. PIV measured velocity field at three cross-sections for closed cavity (upper row) and free surface case (bottom row); steady state; Ra=37,500, Pr=8,700.

Over 20 different experimental runs have been performed covering different flow regimes (Ra = 22,500, 37,500 and 395,000) and upper boundary conditions (solid and free), and for both transient and steady flow. To get a better view of the flow structure, the steady results were collected at up to 10 different vertical and horizontal cross-sections. To explore possible effects of surface tension, a series of free surface runs were repeated adding a small amount of surface active agent (Fotonal) to the liquid. Finally, runs were conducted using pure water ($Ra = 2\times10^6$), observing mainly the flow pattern appearing at the free surface.

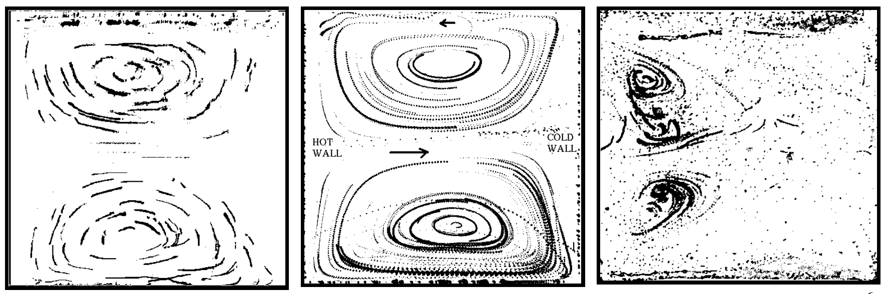
A common feature of all transient experiments is that the flow rising up the hot wall reaches the top of the cell during the first few minutes. The initially dominant conduction, characterised by vertical isotherms, is gradually replaced by convection. The velocity field at the centre vertical plane exhibits strong asymmetry, with the recirculation centre located close to the hot wall. After about 5 minutes the effect of convection becomes evident, accelerating fluid far from the hot wall. This behaviour was only slightly affected by the presence of the free surface. Similar images of the flow pattern are observed for the onset of convection for the closed cavity[1]. The presence of a free surface increases both the vertical and horizontal velocity components near the top hot corner of the cavity, compared with the solid top wall. The temperature distribution indicates that with a free surface, hot liquid penetrates much further along the surface towards the cold wall. Hence, the maximum temperature gradient at the top cold corner increases. Initially, faster penetration of hot liquid creates a stagnation region in the lower part of the cavity. It takes almost 60 minutes for the whole pattern of isotherms to drift down to the steady state configuration and fluid circulation fills the whole cavity. Compared with the closed cavity, the initial development of the free-surface temperature fields is faster, but the whole time scale for the onset of convection does not change much.

Once steady flow is established in the cavity, the flow structure shows a large spiralling motion, with the outer spiral transporting liquid from the centre symmetry plane to the side walls and back to the core along an inner spiral. Both particle visualisation and velocity fields measured at horizontal cross-sections indicate the appearance of this cross-flow. The presence of a free surface slightly changes this structure, increasing the hot-cold asymmetry of the flow (Fig. 1). In addition, a very slow circulation appears at the surface, consisting of a flow along the centre of the surface from the hot towards the cold wall and a reverse flow along the edges of the surface (Fig. 2). This surface flow appears to be decoupled from the main circulation pattern in the cavity.

Compared with experimental results for the closed cavity, the main features appearing due to the free surface are:

- The maximum horizontal velocity, which is located close to the top of the cavity, is almost 50% higher; the maximum vertical velocity, which is located close to the hot wall, remains almost unchanged.
- The onset of convection leads to an initial flow stratification, with the main circulation limited to the upper part of the cavity. The stratification gradually disappears.

- The interaction of the non-isothermal side walls with the main flow slightly modifies the flow structure in free-surface conditions, shifting the centre of circulation closer to the cold wall.
- Most significantly, a horizontal, apparently two-dimensional recirculation appears in a thin surface layer. A surfactant added to the fluid had no effect: the main flow structure and the anomalous recirculation at the surface were unchanged.



(a) *Ra*=37,500 (b) *Ra*=395,000 (c) *Ra*=$2 \cdot 10^6$

Figure 2. Steady state free surface flow pattern (y=0.9); (a) - 24 superimposed images taken every 20s, (b) - 100 images taken every 20s, (c) - 25 images taken every 20s.

These experimental results were investigated numerically for both closed and open surface conditions. Several computational runs were performed with varying thermal boundary conditions at the non-isothermal boundaries (adiabatic; with an expected convective heat flux; and with overestimated and underestimated values), with different fluid properties (Boussinesq and variable), and for all three experimental flow regimes (*Ra* = 22,500, 37,500 and 395,000). The aim was to find flow parameters crucial for the observed flow structures, and to examine hypotheses for possible factors responsible for the recirculation observed at the surface.

Overall agreement between the calculated and measured general flow characteristics is satisfactory. The temperature and velocity fields are numerically well reproduced for both free surface and solid wall conditions. Numerical particle tracking, which is the best tool for investigating 3D flow fields, does not indicate any apparent influence of the free boundary conditions on the spiralling flow structure. As with the

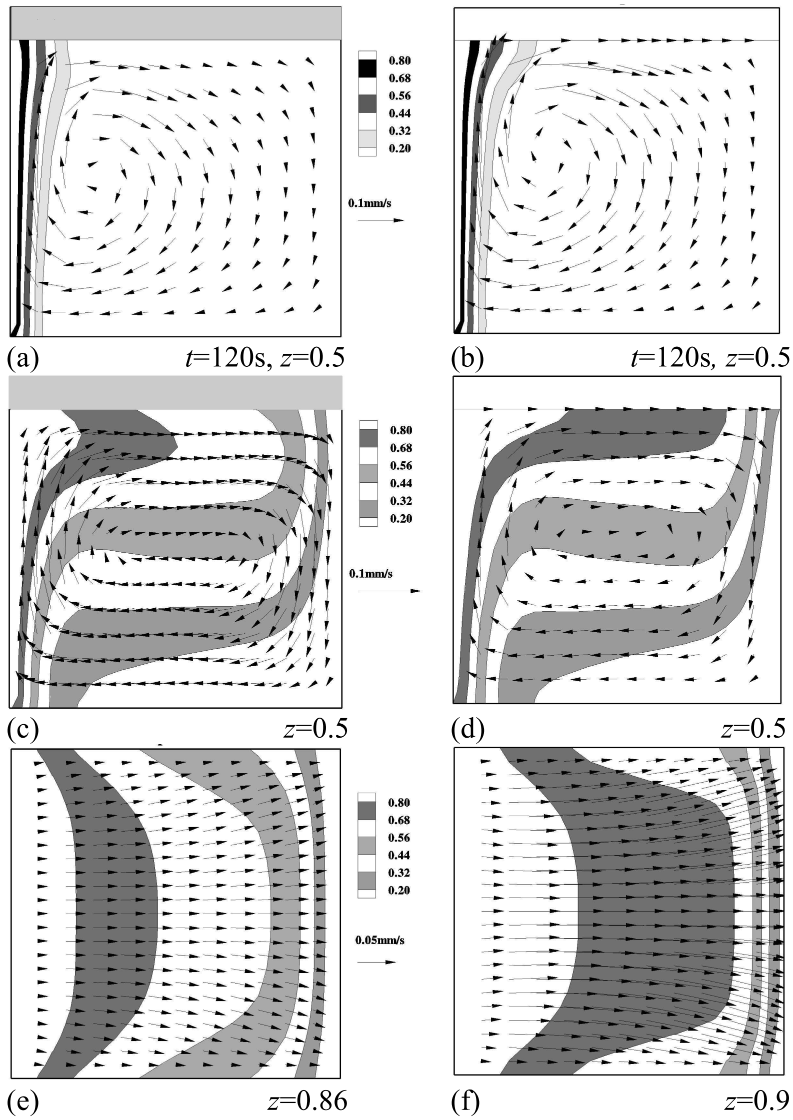


Figure 3. Computed temperature and velocity fields for closed box (left) and free surface flow (right). Transient state (top), steady state centre plane (middle) and horizontal top plane (bottom); Ra=37,500, Pr=8,700.

closed cavity observations (Hiller et al.[3]), both bending of the inner spiral, and merging of spirals are visible. A small asymmetry appearing in the free surface flow (comp. Fig. 1b,e) is not reproduced by the numerical model. This effect can be accounted for through changes in the heat balance for the thick walls of the cavity, when hot fluid is no longer touching the lid. Our previous investigations[3,4] showed the high sensitivity of 3D flow structures to small changes of thermal boundary conditions at non-isothermal walls. To investigate such effects the numerical model needs to solve the conjugate heat conduction problem for the solid walls.

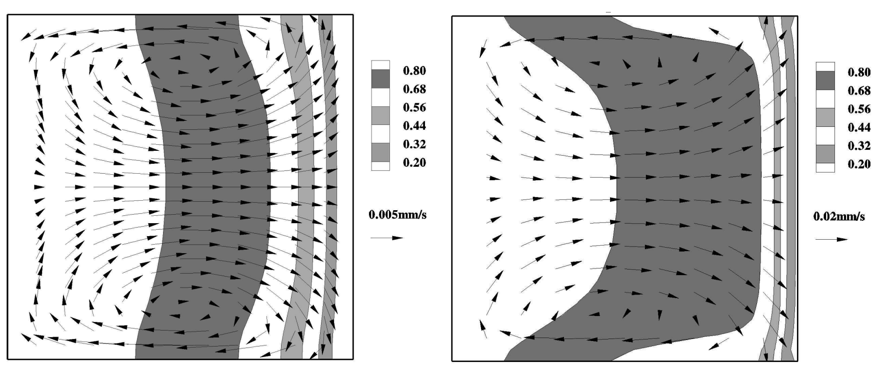
However, the recirculating flow observed at the surface could not be reproduced or explained by these calculations. As noted above, we neglected thermocapillarity. This force would be directed along the main flow direction and could only serve to slightly speed up the surface flow. With standard kinematic boundary conditions at the surface, neither strong variations of the heat flux at the non-isothermal boundaries (even beyond realistic values), nor a relatively large variation of fluid viscosity could produce the observed recirculation. Through numerical experimentation it also became clear that neither a small surface curvature (meniscus) nor a dynamic coupling between the air gap and the fluid below could induce forces strong enough to reverse the flow at the surface.

5.1 A new surface layer model

The effect of surface tension on the hydrodynamics appears in a change of the boundary conditions for the tangential and normal components of the surface stress tensor. The idealised model of a liquid interface includes several simplifications, not necessarily justified by physics. Knowing that the liquid is composed of molecules in motion kept close to each other by van der Waals forces of attraction, the liquid surface can be seen as a region of non-equalised interactions. The energy of a surface molecule is therefore higher than that of a molecule in the bulk fluid, and additional energy must be used to move a molecule from the interior to the surface. Modification of inter-molecular forces may extend over several molecular layers adjacent to the surface. Due to their different mobility, the macroscopic flow can also be affected (*e.g.* the phenomenon known as surface viscosity). In most practical applications these effects can be completely neglected.

However, to find a proper interpretation for the surface effects we observed, we must extend the standard model of the liquid surface. According to our hypothesis, the slow, low energy bulk circulation in the cavity interacts with a free surface layer of finite thickness and access energy. Hence, only molecules of sufficient kinetic energy may leave or enter the free surface. Whether this surface layer is only several molecules thick or whether it extends to a macro-layer is not known. It was not visible when viewed from the side of the cavity, but was clearly visible from above.

To explore this model, some numerical computations were performed in which the free surface was represented as an impermeable but mobile layer three nodes thick in which the vertical velocity component u_y was artificially suppressed. The bulk 3D motion below the surface drives a 2D motion in the layer. To simulate the thin but finite thickness of this layer, a non-uniform mesh was used with the three points located at dimensionless distances of 0.00167, 0.00333 and 0.006 below the surface (compared with a cavity depth of 0.9). These dimensions are far from the molecular scale and were chosen only to ensure code stability. The kinematic boundary conditions on the free surface, viz., $u_y = \partial u_x /\partial y = \partial u_z /\partial y = 0$, were applied at all nodes in the layer.



(a) *Ra*=22,500

(b) *Ra*=395,000

Figure 4. Temperature and velocity fields computed using the proposed surface layer model. Top plane (*y*=0.9), steady flow.

Figure 4 shows the computed temperature and velocity fields for the surface layer obtained for two flow conditions (*Ra* = 22,500 and 395,000). In both cases the recirculation is very similar to that in the experiments. The mean flow velocity at the surface is an order of magnitude lower that that in the bulk flow. The non-dimensional height of the surface layer in the computational model is 0.006. Although this is physically unrealistically thick, it is small enough to be both thermally and dynamically negligible for the bulk flow field.

6 Concluding remarks

Both the experimental and numerical investigations indicate that the free surface boundary condition has a relatively small effect on the bulk flow structure. The main differences appear for both the velocity and temperature fields in the upper part of the cavity. The numerical models proved their ability to describe most of the experimental features. The main result of this work is the empirical evidence of a two-dimensional recirculating flow present at the free surface. The physical interpretation of this flow is rather intuitive, exploring the idea of a molecular surface layer. The structure and properties of surface monolayers are functions of the molecular arrangement and their interactions. Surface active agents may significantly modify the physical structure of the surface and hence its mechanical properties. However, our experiments showed that there was no effect on the anomalous surface flow. Absent any physical explanation, the observed surface effect is nevertheless reproducible in our experiments and presumably can be found in many other free surface flow configurations. Hence, our present attempt to explore this phenomenon through a computational model of a thin surface layer may be useful. Our simple model does not pretend to describe the physics at the surface. For this purpose, a reliable description of intermolecular forces for a given physical surface structure must be known *a priori*, to provide values for the surface access energy, surface viscosity, and molecular depth of the layer. Our present aim is rather to indicate that the proper interpretation of some free surface flow patterns may need a reassessment of the standard boundary conditions.

References

[1]Hiller, W.J., Koch, St., Kowalewski, T.A., Stella, F., Onset of natural convection in a cube, *Int. J. Heat Mass Transfer*, **36**, pp. 3251-63, 1993.

[2]Mallinson, G.D. and de Vahl Davis, G., The method of the false transient for the solution of coupled elliptic equations, *J. Comp. Phys.* **12**, pp. 435-461, 1973.

[3]Hiller, W.J., Koch, St., Kowalewski, T.A., de Vahl Davis, G., Behnia, M., Experimental and numerical investigation of natural convection in a cube with two heated side walls, *Topological Fluid Mech.,* eds. H.K Moffat & A. Tsinober, pp. 771-726, CUP, Cambridge 1990.

[4]Kowalewski T.A., Experimental validation of numerical codes in thermally driven flows, in *Adv. in Comp. Heat Transfer, CHT-97*, Begell House, New York (in press).