International Journal of Thermal Sciences 95 (2015) 47–52

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

journal homepage: www.elsevier.com/locate/ijts

# On the use of stimulated thermocapillary currents and virtual walls as computational tools for natural convection simulation in enclosed spaces

Francisco J. Arias*, Geoffrey T. Parks

*Department of Engineering, University of Cambridge, Trumpington Street, Cambridge CB2 1PZ, United Kingdom*

A R T I C L E I N F O

*Article history:*
Received 18 April 2014
Received in revised form
17 November 2014
Accepted 11 March 2015
Available online 24 April 2015

*Keywords:*
Natural convection
Effective thermal conductivity
Thermocapillary convection
Computational fluid dynamics (CFD)

A B S T R A C T

A new, alternative approach is proposed for natural convection simulation by means of stimulated thermocapillary currents created by virtual walls. In contrast to the well-known effective thermal conductivity model, in the proposed approach it is the mass motion due to the convective currents which is intended to be simulated and the heat flux is a consequence of such flows. As a result, no a priori knowledge of the Nusselt number is needed and thus the approach is more suitable for complex geometries. Utilizing a simplified physical model and the definition of hydraulic diameter, a generalized expression for enclosed geometries is derived which offers thermal engineers a powerful analysis tool that can use virtual walls with an associated fictitious Marangoni stress for pre-screening and estimation of Nusselt numbers.



## 1. Introduction

Physical enclosures are frequently encountered in practice, and heat transfer through them is of practical interest [1–6]. However, heat transfer in enclosed spaces is complicated by the fact that the fluid in the enclosure, in general, does not remain stationary. For example, in a vertical enclosure, the fluid adjacent to the hotter surface rises and that adjacent to the cooler one falls, establishing rotatory motion (natural convection) within the enclosure that enhances heat transfer. Over the years, several methods have been proposed for the simulation of natural convection, and among them the most popular extension is the use of an effective diffusivity term (effective thermal conductivity) to convert the effects of convection into pure conduction [7,8]. However, although such an approach offers a viable representation of heat transfer by natural currents, it has an important associated disadvantage. It is therefore worthwhile to briefly review the traditional effective diffusivity method to identify this weakness and then propose an alternative approach which can be used to overcome it.

### A. The effective thermal conductivity model

When the Nusselt number **Nu** is known, the rate of heat transfer $\dot{Q}$ through an enclosure can be determined from Ref. [9]:

$$\dot{Q} = hA_s(T_1 - T_2) = \kappa \mathbf{Nu} A_s \frac{T_1 - T_2}{L} \tag{1}$$

where $T_1 - T_2$ is the temperature difference over a distance $L$, $A_s$ is the area, and the heat transfer coefficient $h$ is related to the thermal conductivity $\kappa$ through the definition of the Nusselt number:

$$\mathbf{Nu} = \frac{hL}{\kappa} \tag{2}$$

The rate of steady heat conduction $\dot{Q}_{cond}$ across a layer of thickness $L_c$, area $A_s$, and thermal conductivity $\kappa$ is:

$$\dot{Q}_{\mathrm{cond}} = \kappa A_s \frac{T_1 - T_2}{L_c} \tag{3}$$

where $T_1$ and $T_2$ are the temperatures on the two sides of the layer.

By comparing Eqs. (1) and (3) it can be seen that the convective heat transfer in an enclosure is analogous to heat conduction across the fluid layer in the enclosure, provided that the thermal conductivity $\kappa$ is replaced by $\kappa\mathbf{Nu}$ [9]. In other words, the effective thermal conductivity model says: *the fluid in an enclosure behaves*

* Corresponding author. Tel.: +44 32 14 33 21 94.
*E-mail address:* fja30@cam.ac.uk (F.J. Arias).

http://dx.doi.org/10.1016/j.ijthermalsci.2015.03.013
1290-0729/

*like a fluid the thermal conductivity of which is* $\kappa$***Nu*** *as a result of convection currents*. That is:

$$\kappa_{\text{eff}} = \kappa \cdot \mathbf{Nu} \tag{4}$$

However, as is readily apparent, this effective conductivity model has a serious weakness which limits its applicability in certain circumstances, namely: *it assumes* a priori *knowledge of the Nusselt number.*

This weakness is somewhat paradoxical, because, in many instances, it is precisely the value of the Nusselt number that is sought. So, for example, in attempting to apply the effective thermal conductivity model in an application with complex or unusual geometry, say, in the field of microelectronics, where it might be very difficult to find in handbooks or the available literature a standard Nusselt number correlation for the specific design under consideration, the thermal engineer will need to exercise their judgement in deciding on the most appropriate correlation in order to define the effective thermal conductivity in Eq. (4).

In order to address this problem, in this paper, an alternative method is proposed that makes use of artificially induced thermocapillary flow associated with virtual walls which will promote a mass flow similar to that produced by gravity. The proposed method could be a powerful pre-screening tool for thermal engineers by which to obtain preliminary information about the Nusselt number correlation, and thus to define an effective thermal conductivity. In the next section, we discuss briefly the fundamentals of the proposed approach.

B. Thermocapillary convection as a simulated induced flow

Marangoni convection occurs when the surface tension of an interface depends on the concentration of a species or on the temperature distribution. In the case of temperature dependence, the Marangoni effect is also called thermocapillary convection. The Marangoni effect is of primary importance in the fields of welding, crystal growth and electron beam melting of metals. For this study it is sufficient to know that as a result of this phenomenon a shear stress is developed in the interface which is caused by the variation of surface tension. The interested reader is referred to reference [10] for further information about this phenomenon and supporting theory.

Within the framework of this kind of convection, a shear stress $\tau_\sigma$, which is applied at the wall, is developed. Its value is given by:

$$\tau_\sigma = \nabla_T \sigma \cdot \nabla_s T \tag{5}$$

where $\nabla_T \sigma = d\sigma/dT$ is the surface tension gradient with respect to temperature, and $\nabla_s T$ is the surface temperature gradient.

Two aspects of the relationship in Eq. (5) suggest this as a potential application for one-dimensional/two-dimensional natural convection modelling: first, this shear stress is a one-dimensional effect; and second, the effect is driven by a temperature gradient.

## 2. Model description

Although, in order to simulate a fluid flow (in this case a thermocapillary flow) in full detail, it is necessary to describe the associated physics in mathematical terms through conservation principles, with the use of nonlinear partial or ordinary differential equations to express these principles [11,12], for preliminary assessment purposes a simple model based on momentum balance is preferable and may be sufficient.

Let us consider a control volume, as depicted in Fig. 1, in order to establish a simplified mathematical model that will allow us to find a suitable expression for the induction of thermocapillary currents in a manner similar to gravitational currents, at least from the point of view of mass transport.

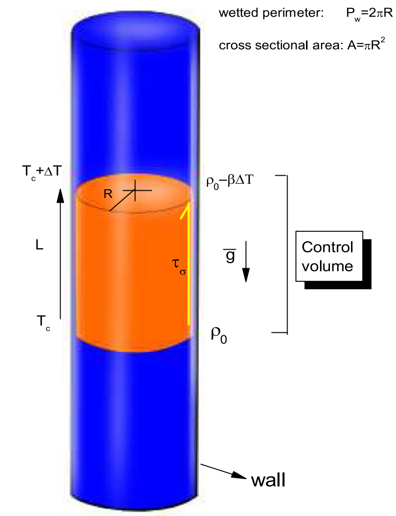


**Fig. 1.** Control volume used for calculations.

A. Derivation

First, to analyse flow through the control volume, we need to establish the forces acting on the control volume. The pressure force pushing the liquid through the tube due to the buoyancy forces is given by the change in pressure multiplied by the area:

$$F_g = \rho_o \beta g(\Delta T) \cdot A \cdot L \tag{6}$$

where $\rho_o$ is the average density of fluid, $\beta$ is the volumetric coefficient of thermal expansion, $g$ is the acceleration due to gravity, $\Delta T$ is the increase in temperature over the length of the control volume, $A$ is the cross-sectional area, i.e., $A = \pi R^2$, and $L$ is the length of the control volume.

We can define a (fictitious) Marangoni stress associated with the walls as:

$$F_\sigma = \tau_\sigma \cdot P_W \cdot L \tag{7}$$

where $\tau_\sigma$ is the Marangoni stress defined previously in Eq. (5), $P_W$ is the wetted perimeter, i.e., $P_W = 2\pi R$, and $L$ is again the length of the control volume.

Thus, if we want our induced artificial thermocapillary flow to “replace” the buoyancy-induced flow, this can be accomplished by arranging that:

$$F_\sigma = F_g \tag{8}$$

or, taking into account Eqs. (6) and (7):

$$\tau_\sigma = \rho_o \beta g \Delta T \frac{A}{P_W} \tag{9}$$

A hydraulic diameter $D_h$ may be defined as [9]:

$$D_h = \frac{4A}{P_W} \tag{10}$$

thus enabling Eq. (9) to be rewritten more compactly as:

$$\tau_\sigma = \rho_o \beta g \Delta T \frac{D_h}{4} \tag{11}$$

Eq. (11) thus allows us to calculate the equivalent fictitious Marangoni stress for any enclosed geometry by means of the calculation of its hydraulic diameter.