2002

A CFD Model For Buoyancy Driven Flows Inside Refrigerated Cabinets and Freezers

C. J. L. Hermes
Whirlpool Corporation

M. E. Marques
Whirlpool Corporation

C. Melo
Federal University of Santa Catarina

C. O. R. Negrao
Federal Center of Technological Education of Parana

A CFD MODEL FOR BUOYANCY DRIVEN FLOWS INSIDE REFRIGERATED CABINETS AND FREEZERS

Christian J. L. Hermes
Marco E. Marques
Multibrás S.A. Eletrodomésticos, Whirlpool Corporation
7200 Dona Francisca St., 89219-900, Joinville, SC, Brazil, +55 47 441 49 83
chermes@multibras.com.br, marcom@multibras.com.br

Cláudio Melo
Mechanical Engineering Department, Federal University of Santa Catarina
P.O. Box 476, 88040-900, Florianópolis, SC, Brazil, +55 48 234 56 91, melo@nrva.ufsc.br

Cezar O. R. Negrão
Academic Department of Mechanics, Federal Center of Technological Education of Paraná
3165 Sete de Setembro Av., 80230-901, Curitiba, PR, Brazil, +55 41 310 46 58, negrao@cefetpr.br

ABSTRACT

The streamfunction – vorticity formulation is implemented in a CFD code for the analysis of air flow patterns and temperature distributions in closed cavities, more specifically refrigerated cabinets and freezers. The model is based on the mass, momentum and energy conservation principles. The flow is treated as incompressible and the buoyancy terms are modeled according to the so-called Boussinesq approximation. The finite volume method and an iterative procedure are employed to discretise and solve the governing set of partial differential equations. The presented methodology is simpler than the traditional approach based on the primitive variables since the mass conservation principle is established at each iteration and there is no need for any additional treatment of the pressure – velocity coupling. The program performance is evaluated by comparisons with numerical benchmark solutions of classical buoyancy driven cavity flows found in the literature. Simulation results for a one door all-refrigerator are presented and discussed. Comparisons with experimental data are also reported.

NOMENCLATURE

u, v : velocity components [m/s]
T : temperature [K]
T_o : reference temperature [K]
p : pressure [Pa]
k : thermal conductivity [W/m.K]
q : heat flux [W/m^2]
g : gravity acceleration [m/s^2]

Greek symbols
α : thermal difusivity [m^2/s]
ν : kinematic difusivity [m^2/s]
β : thermal expansion coefficient [K^{-1}]
ρ : specific mass [kg/m^3]
ψ : streamfunction
ω : vorticity

INTRODUCTION

Knowledge of the air flow patterns inside refrigerated cabinets is essential for the proper design of household refrigerators, specially in cases where natural convection is the only driver for the air flow. Both experimental and numerical approaches are often used for these studies. The latter is the focus of this work.

Very few works dealing with heat and fluid flow inside refrigerated cabinets can be found in the open literature. Dirik et al. (1996) studied the 3D natural convection air flow inside an all-refrigerator using a commercial CFD package. Deschamps et al. (1999) developed a model based on the commercial package FLUENT to analyze the 3D air flow inside an all-refrigerator, considering both turbulence and thermal radiation effects. Cortela et al. (2001) analyzed the forced convection air flow inside a commercial open display cabinet using a 2D CFD code based on the streamfunction – vorticity formulation that incorporates a LES (Large Eddy Simulation) turbulence model.

As a contribution to this field, this research effort is intended to develop a CFD code to be used in the modeling of a vapor compression refrigeration system coupled with a refrigerated cabinet as a whole (Hermes, 2000). This paper focuses only in the CFD code development.

The present model uses a methodology quite close to the one proposed by Cortela et al. (2001). The model equations are based on the streamfunction – vorticity formulation and on the thermal energy conservation principle. The buoyancy terms are modeled considering a constant-property Boussinesq fluid and the resulting set of equations is solved by the finite volume technique. The current model, although restricted to 2D applications, establishes a mass balance at each control volume, making the numerical algorithm faster and more robust.

The mathematical model and the numerical techniques are firstly described. Secondly comparative analyses with similar works found in the literature are performed. Next comparisons with experimental data are performed in order to validate the proposed model. Finally the velocity and temperature distributions inside an empty household refrigerator cabinet are simulated and discussed.

MATHEMATICAL FORMULATION

Governing Equations

The phenomena associated with natural convection inside two-dimensional enclosures are governed by the mass, Navier-Stokes and thermal energy conservation equations, given respectively by:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$\frac{\partial u}{\partial t}+\frac{\partial}{\partial x}(uu)+\frac{\partial}{\partial y}(vu)=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t}+\frac{\partial}{\partial x}(uv)+\frac{\partial}{\partial y}(vv)=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)-g[1-\beta(T-T_o)] \tag{3}$$

$$\frac{\partial T}{\partial t}+\frac{\partial}{\partial x}(uT)+\frac{\partial}{\partial y}(vT)=\alpha\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

The following simplifying assumptions were adopted: incompressible laminar flow, negligible viscous dissipation, Newtonian and constant-property Boussinesq fluid. The first two assumptions are, in fact, a consequence of the Boussinesq approximation (see Kundu, 1990). For more information on the physics of natural convection inside closed cavities please see the works of Ostrach (1988) and Bejan (1995).

The pressure, the temperature and the scalar velocity components, *u* and *v,* are the problem unknowns. In the absence of an evolutionary pressure equation, an algorithm for the pressure-velocity coupling must be employed (Patankar, 1980).

An alternative formulation for this type of flow is the one based on the concepts of streamfunction, ψ, and vorticity, ω. For 2D problems, the former is defined as a line tangent to the velocity vector and the latter as the curl of the velocity field. From these definitions and through a series of algebraic operations, the following equations can then be derived:

$$\frac{\partial \omega}{\partial t}+\frac{\partial}{\partial x}(u\omega)+\frac{\partial}{\partial y}(v\omega)=\nu\left(\frac{\partial^2 \omega}{\partial x^2}+\frac{\partial^2 \omega}{\partial y^2}\right)+g\beta\frac{\partial T}{\partial x} \tag{5}$$

$$\frac{\partial^2 \psi}{\partial x^2}+\frac{\partial^2 \psi}{\partial y^2}+\omega=0 \tag{6}$$

This formulation offers some advantages when compared to the traditional formulation based on the primitive variables. The mass conservation principle, for example, is established for any streamfunction value. Therefore the required number of equations is reduced from four (u, v, p e T) to three (ψ, ω, T). An additional advantage is the elimination of the pressure terms in the Navier-Stokes equations. The difficulties in identifying the boundary conditions, specially for open boundaries problems, is one of the disadvantages of this methodology.

Boundary Conditions

Equations (4) to (6) are elliptical both in the x and in y directions and therefore two boundary conditions are required in each direction. The so-called non-slip boundary conditions were used for the velocities at impermeable static surfaces. For the streamfunction – vorticity formulation, the non-slip boundary condition can be expressed as (Roache, 1998):

$$\psi_s=0 \tag{7}$$

$$\omega_s=-2\frac{\psi|_{s+1}}{\Delta n^2} \tag{8}$$

An additional equation for the temperature distribution along the cabinet walls was also required:

$$\frac{\partial T_w}{\partial t}=\alpha_w\left(\frac{\partial^2 T_w}{\partial x^2}+\frac{\partial^2 T_w}{\partial y^2}\right) \tag{9}$$

where T_w is the cabinet wall temperature [K] and α_w is the thermal diffusivity of the insulating foam [m^2/s].

Equations (4) and (9) may be coupled by the following equation:

$$-k_w\left.\frac{\partial T}{\partial n}\right|_{w,i}+q_{rad}=q_{w,i} \tag{10}$$

where k_w is the wall thermal conductivity [W/m.K] and q_{rad} is the radiative heat flux between the internal surfaces of the cabinet and the evaporator [W/m^2]. At the external surfaces, the boundary conditions were obtained by the well known Newton's law of cooling:

$$q_{w,o}=U\left(T_\infty-T_{w,o}\right) \tag{11}$$

where $T_{w,o}$ is the cabinet external wall temperature [K], T_∞ the surrounding air temperature (43°C), and U the overall heat transfer coefficient (7.0W/m^2K). The radiative heat flux was calculated using a variation of the model presented by Incropera & DeWitt (1990) for radiative heat exchanges between black surfaces. The evaporator surface temperature was taken as a prescribed boundary condition (–20°C). The heat gain through the door gasket was not taken into account.

NUMERICAL METHODOLOGY

The resulting set of differential equations was solved through a finite volume methodology (Patankar, 1980). Such method is based on the discretization of the domain into non-matching small control volumes where the relevant variables are conserved. Expressing the differential equations for the scalar variables ψ, ω and T through an equivalent equation for the advective-diffusive transport of a generic scalar unknown ϕ, yields,

$$\frac{\partial\phi}{\partial t}+\frac{\partial}{\partial x}(u\phi)+\frac{\partial}{\partial y}(v\phi)=\Gamma\left(\frac{\partial^2\phi}{\partial x^2}+\frac{\partial^2\phi}{\partial y^2}\right)+S \tag{12}$$

The above expression when applied to the typical cartesian control volume of the discretized domain shown in figure 1, yields:

$$A_P\phi_P-\sum A_{NB}\phi_{NB}=B \tag{13}$$

In the above system of linear equations the *NB* subscript represents the neighborhood control volumes (*N*, *S*, *E*, *W*), Γ the diffusivity of the transport property ϕ [m^2/s], and *S* the source term of equation (12). The resulting set of algebraic equations was grouped as shown in figure 2 and solved simultaneously through the conjugated gradient method (Press et al., 1995).

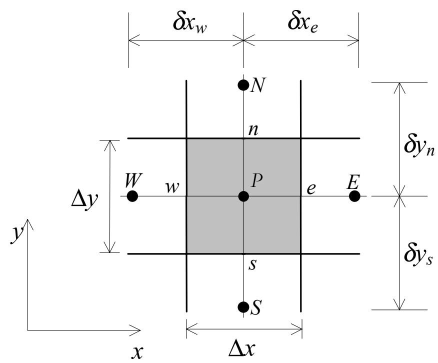


Figure 1. Typical control volume of the discretized domain.

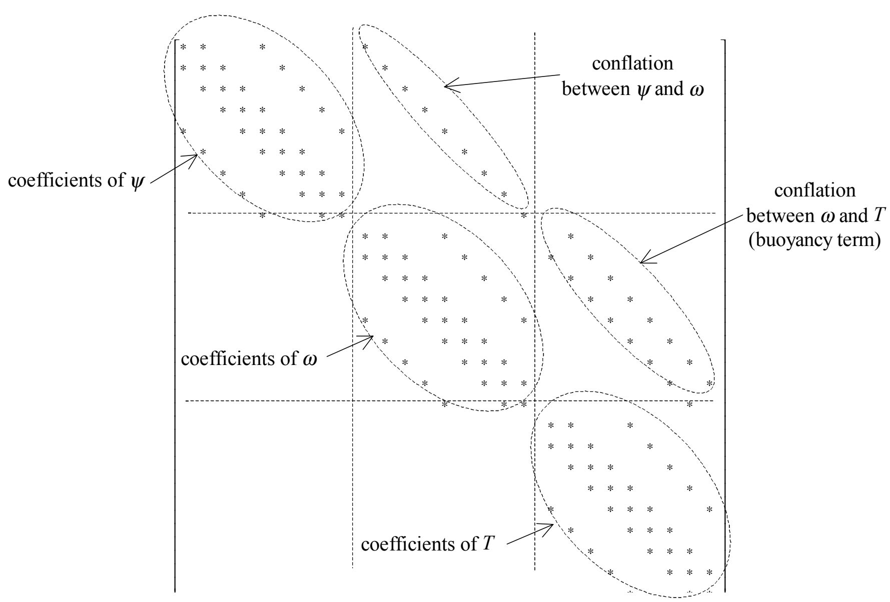


Figure 2. Overall matrix of coefficients (example for a mesh with 3x3 control volumes).

The simultaneous solution process implies in the storage of the transient terms in the source terms of the algebraic equations. This suggests that an artificial transient term may be employed in equation (6) to improve the convergence process,

$$\frac{\partial \psi}{\partial t} = \frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} + \omega \qquad (14)$$

It should be mentioned that at steady-state conditions, the artificial transient term will vanish and equation (14) will become identical to equation (6). It should also be noted that such procedure is totally equivalent to the use of sub-relaxation, and that it is only indicated for steady-state analysis.

The discretization process employed a non-uniform cartesian mesh with a co-located grid. Additional cells were used in the external boundary of the domain. The mesh was generated algebraically from the equations proposed by Wood (1996). The time and space discretization adopted a fully implicit and a power-law interpolation scheme, respectively (Patankar, 1980). The developed algorithm, based on three iterative loops, is described below:

- An external loop for the actual or pseudo transients, with time-steps of the order of 0.1s. The equations are considered solved when the sum of all relative errors is lower than 10^{-6} (pseudo transient) or when the length of time prescribed for the simulation is reached (actual transient).
- An intermediate loop for the update of the matrix of coefficients. The convergence is achieved when the sum of all relative errors is lower than 10^{-6} (actual transient) or when a prescribed number of iterations is reached (pseudo transient).
- An internal loop for the simultaneous solution of the set of linear equations. The convergence is achieved when the sum of all normalized residues is lower than 10^{-6}. After that the scalar components of the velocity vector are calculated from the streamfunctions patterns through a CDS (Central Difference Scheme) approximation.

PROGRAM EVALUATION

The program performance was evaluated by comparisons with numerical benchmark problems proposed in the literature. Two typical problems on square cavities flows were considered. The first problem is the one presented by Ghia et al. (1982), where the flow is induced by shear stresses in a square cavity through an upper horizontal plate. The second problem is the one presented by Hortmann et al. (1990), that is essentially a natural convection flow inside a square cavity formed by two insulated horizontal walls and two uniform but at different temperature vertical walls. In both cases an excellent level of agreement between the model predictions and the published results was achieved.

SIMULATION RESULTS

A schematic representation of the refrigerator investigated in this work is shown in figure 3. Besides the geometric data only the surrounding air temperature (43°C) and the evaporator temperature (–20°C) were supplied to the computer code as input data. The inner liner surfaces were treated as flat plates, and the shelves were not considered in the present analysis. Mesh independent results were obtained for meshes with at least 1800 (30x60) control volumes. Although dynamic simulations can also be performed the current analysis focuses only in the steady-state results.

Table 1 shows a comparison between the numerical and experimental air temperature values for a surrounding (test room) air temperature of 43°C. As one can see the experimental data were quite well reproduced by the computer code with a maximum absolute deviation of 1.7°C.

It has to be stressed that such a level of agreement was only achieved because the boundary layer dominant flow inside the cabinet implies in weak convection and strong diffusion effects in the points located far away from the walls, mainly in the center points of the cabinet where the probes (thermocouples) were positioned.

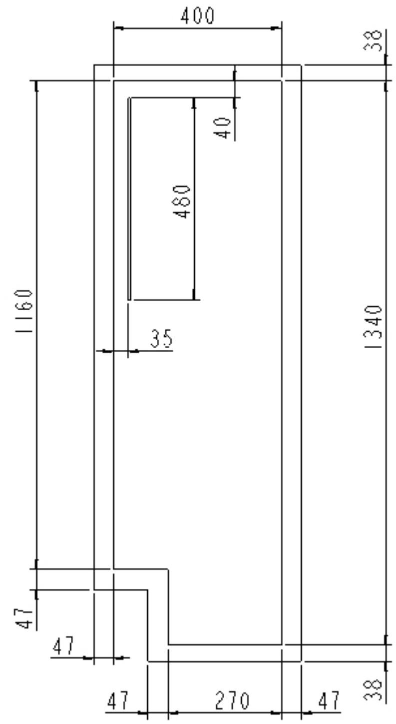


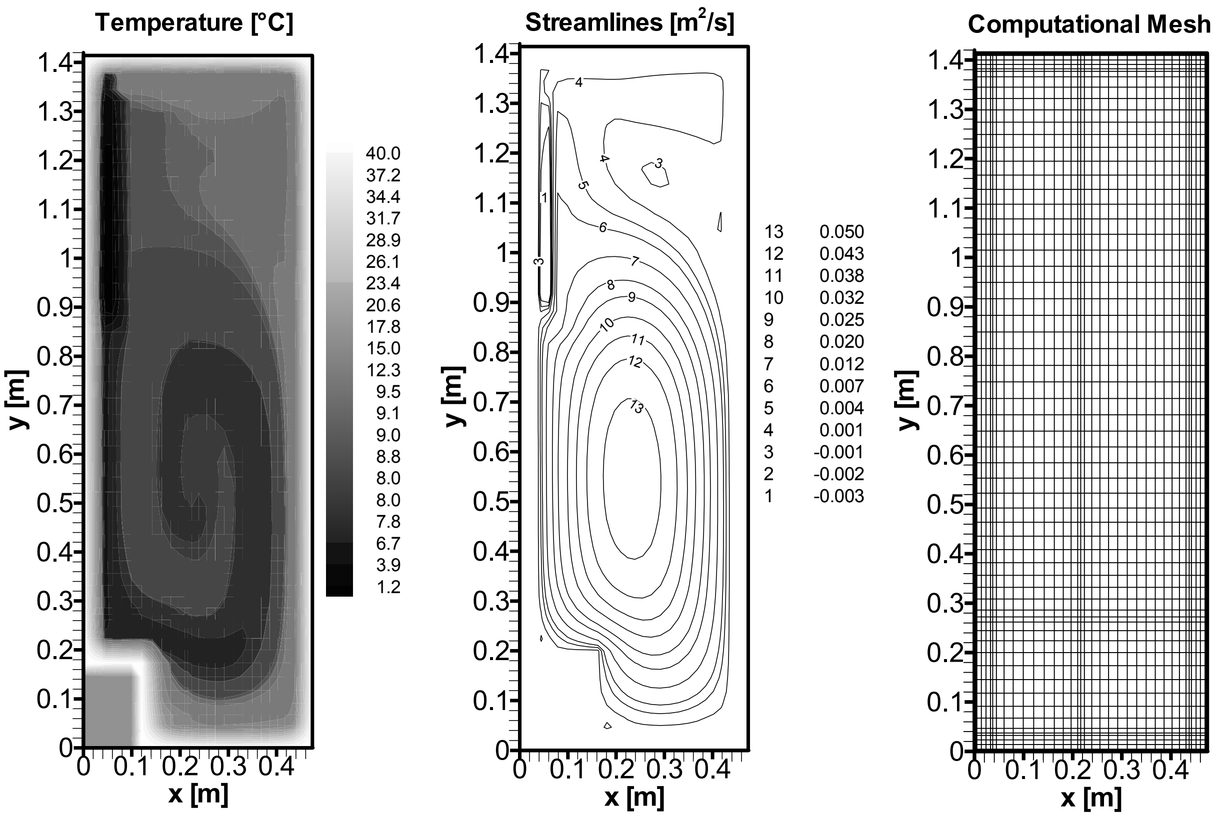
Figure 3. Vertical mid plane of the refrigerator under analysis.

Table 1. Simulated versus experimental data.

Probe location	Results		
x-y coordinates [mm]	Numerical [°C]	Experimental [°C]	Difference [°C]
237 - 1117	9.1	7.4	1.7
237 - 937	8.4	7.6	0.8
237 - 677	8.0	7.5	0.5
237 - 447	7.0	5.6	1.4
302 - 147	7.9	6.4	1.5

Figure 4 shows the steady-state temperature distributions, the streamlines and the computational mesh on the vertical mid plane of the refrigerator. It can be seen that the air is cooled by the evaporator and moves downwards, almost vertically. The lowest air temperatures are found next to the back wall and close to the compressor compartment. The warmer air close to the door moves upwards until it is again affected by the evaporator, forming a re-circulation area that prevails in most of the cabinet. A second re-circulation area can be observed in the upper part of the cabinet, and this explains the higher temperature levels in that region. In that area the diffusive effects are predominant due to the low air velocities.

A third re-circulation area can also be observed in the channel formed between the back wall and the evaporator. The air confined in that region does not affect the internal air temperature distribution. On the contrary, the back wall temperature in that region becomes even cooler and this increases the thermal load, degenerating the refrigerator performance. This suggests that attention has to be given to the evaporator positioning and inclination in order to improve the air and temperature distributions inside the refrigerated cabinet.



(a) (b) (c)

Figure 4. Vertical mid plane results: (a) Temperature distribution; (b) Streamlines; (c) Computational mesh.

CONCLUDING REMARKS

A simplified two-dimensional CFD model for predicting the velocity and temperature distributions inside enclosures was presented. The model reproduced typical numerical benchmark solutions of natural convection flows in an exact manner. Comparisons were performed with available experimental data and a reasonable level of agreement was found. A one-door Brazilian refrigerator was simulated and three re-circulation areas were detected.

In recent years, knowledge regarding the air flow behavior inside refrigerated cabinets has become of great importance for engineering design. It is therefore necessary to predict the system performance well in advance and so to optimize the system design by minimizing the energy consumption. This paper presented a numerical model for the heat and fluid flow inside refrigerated cabinets that is a part of a household refrigerator overall simulation computer package. This kind of tool may be used not only to reduce the energy consumption but also to significantly reduce the number of real tests needed.

ACKNOWLEDGMENTS

The authors are grateful to Whirlpool Corporation for sponsoring this research program. The continuous support from the Brazilian Bureau for Scientific and Technological Development (CNP_q) is also duly acknowledged. The authors would like to express theirs thanks to Messrs. L.A.T. Vieira and S.J. Silveira for their support in running the experiments and to Mr. G.H.P. Zattar for his assistance with the drawings.

REFERENCES

Bejan, A., "Convection Heat Transfer", John Wiley & Sons, New York, 1995.

Cortela, G., Manzan, M., Comini, G., "CFD Simulation of Refrigerated Cabinets", International Journal of Refrigeration, Vol.24, pp.250-260, 2001.

Deschamps, C.J., Prata, A.T., Lopes, L.A.D., Schmid, A., "Heat and Fluid Flow Inside a Household Refrigerator Cabinet", 20th IIF/IIR International Congress of Refrigeration, Sydney, Australia, CD-ROM, 1999.

Dirik, E., Iz, H., Aydin, C., "Performance Optimization of a Larder Type Refrigerator Unit Using Computer Aided Analysis Tools", International Refrigeration Conference at Purdue, West-Lafayete, USA, pp.459-463, 1996.

Ghia, U., Ghia, K.N., Shin, C.T., "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method", Journal of Computational Physics, Vol.48, pp.387-411, 1982.

Hermes, C.J.L., "Development of Mathematical Models for Dynamic Simulation of Household Refrigerators", M.Sc. thesis, Federal University of Santa Catarina, Florianopolis, SC, Brazil, 2000.

Hortmann, M., Peric, M., Scheuerer, G., "Finite Volume Multigrid Prediction of Laminar Natural Convection: Benchmark Solutions", International Journal for Numerical Methods in Fluids, Vol.11, pp.189-207, 1990.

Incropera, F.P. & DeWitt, D.P., "Fundamentals of Heat and Mass Transfer", John Wiley & Sons, New York, USA, 1990.

Kundu, P.K., "Fluid Mechanics", Academic Press, San Diego, USA, 1990.

Ostrach, S., "Natural Convection in Enclosures", ASME Journal of Heat Transfer, Vol.110, pp.1175-1190, 1988.

Patankar, S.V., "Numerical Heat Transfer and Fluid Flow", Hemisphere, Washington D.C., USA, 1980.

Press, W.H., Vetterling, W.T., Teukolsky, S.A., Flannery, "Numerical Recipes in Fortran 77: The Art of Scientific Computing", Cambridge University Press, Cambridge, UK, 1995.

Roache, P.J., "Fundamentals of Computational Fluid Dynamics", Hermosa Publishers, Albuquerque, USA, 1998.

Wood, W.A., "Multigrid Approach to Incompressible Viscous Cavity Flow", Technical Memorandum, NASA Langley Research Center, Hampton, VA, USA, 1996.