VENTILATION AND COOLING

18TH ANNUAL AIVC CONFERENCE
ATHENS, GREECE, 23-26 SEPTEMBER, 1997

Use of Computational Fluid Dynamics for Modelling Passive Downdraught Evaporative Cooling

M. A. Mansour[*], M. J. Cook[*], A. H. Taki[+], and K. J. Lomas[*]

[*]Institute of Energy and Sustainable Development, De Montfort University
Leicester, LE1 9BH, UK
Tel: +44 116 257 7417 Fax: +44 116 257 7449
http://www.iesd.dmu.ac.uk

[+]Department of Building Studies
School of the Built Environment, De Montfort University
Leicester, LE1 9BH, UK
Tel: +44 116 257 7048 Fax: +44 116 257 7440

Use of Computational Fluid Dynamics for Modelling Passive Downdraught Evaporative Cooling

M. A. Mansour, M. J. Cook, A. H. Taki, and K. J. Lomas

Synopsis

The air flow in a Passive Downdraught Evaporative Cooling (PDEC) tower has been modelled using a Computational Fluid Dynamics (CFD) code. Water is injected into dry warm air and the interaction between the water and the air is represented using a particle transport model. This models the transfer of mass, momentum and heat between the water particles and the air in addition to predicting individual particle trajectories. The CFD code successfully produced predictions for the air flow in such a cooling system and the results are comparable with those obtained from a one dimensional finite difference model. The CFD results however, provide much more spatial information, in particular, individual particle trajectories. CFD also offers far greater potential for modelling full PDEC systems in which the evaporatively cooled air is delivered to occupied spaces.

List of Symbols

C_p	specific heat (J/kgK)
D	diffusivity of water vapour in air (m^2/s)
d	particle diameter (m)
g	gravity vector (m/s^2)
H	enthalpy (J/kg)
h_{fg}	latent heat of evaporation (J/kg)
k	turbulent kinetic energy (m^2/s^2)
m	particle mass (kg)
Pr	Prandtl number
p	pressure (Pa)
S_ϕ	source term
Sh	Sherwood number
T	temperature (K)
T_G	temperature of continuous phase (K)
T_P	temperature of water particle (K)
t	time (s)
U	velocity vector (m/s)
v_r	relative velocity of the particle and the continuous phase (m/s)
W_c	molecular weight of water vapour (kg/kmol)
W_G	molecular weight of the mixture in the continuous phase (kg/kmol)
X	the ratio of the saturated vapour pressure to the continuous phase vapour pressure at a given temperature
X_G	molar fraction of water vapour in the continuos phase
x	displacement vector (m)
Y_{wv}	mass fraction of water vapour(kg/kg)
Γ_{wv}	molecular diffusion coefficient (kg/ms)
Γ_ϕ	diffusion coefficient for ϕ
ϕ	arbitrary variable
λ	thermal conductivity (W/mK)
μ	dynamic (laminar) viscosity (kg/ms)
μ_T	turbulent viscosity (kg/ms)
μ_{eff}	$\mu + \mu_T$ (kg/ms)
θ	azimuth angle (radians)
ρ	density (kg/m^3)
ρ_{Bref}	buoyancy reference density (kg/m^3)
ρ_p	particle density (kg/m^3)
σ_H	turbulent Prandtl numbers for H
ζ	bulk viscosity (kg/ms)

1. Introduction

Passive Downdraught Evaporative Cooling (PDEC) is an energy efficient method of producing cool air in hot dry climates. The process involves injection of a very fine mist of water particles, produced by micronisers, into a warm dry air stream. As the water evaporates, the air temperature decreases by an amount dependent on the amount of water which is evaporated. This cooled air can then be delivered to occupied spaces. A PDEC system can be readily divided into three distinct zones: (i) a wind catcher; (ii) a cool air production zone (or evaporation zone) where water droplets are sprayed into the air stream; and (iii) a region in which the cooled air is delivered to the occupied spaces [1].. It is the modelling of zone two using computational fluid dynamics (CFD) that is the subject of this paper.

The work is part of a three year research project which began in January 1996 under the EC Joule programme [1]. It is a multi-disciplinary project involving architects: Brian Ford & Associates and Mario Cucinella Architects; building physicists and simulation experts at De Montfort University and the University of Malaga; monitoring experts at the Conphoebus Institute in Sicily, and microniser and control specialists Microlide SA. The objective is to study the application of PDEC systems to non-domestic buildings.

In this paper CFD simulations of evaporative cooling is discussed. A very simple, two dimensional case has been considered in which water is injected, evenly distributed, along a horizontal line. The total flow rate is set equal to that from a single (typical) microniser. In order to gain confidence in the CFD predictions, the results are compared with a one dimensional 'tower model' of Rodríguez et al. [2,3] and Alvarez et al [4].

2. Modelling the Evaporation Zone in CFD

2.1 The CFD package

The CFD package used for this work is CFX-F3D [5], version 4.1. This a multiblock code in which geometries are defined using one or more topologically rectangular blocks. Each block is then covered with a mesh and the governing equations solved using the finite volume method on a co-located grid [6].

2.2 Modelling the Evaporation Zone

A particle transport model was used to represent the evaporation zone. In this model, water droplets are considered as a source of mass, momentum and energy in the continuous phase. The model begins by solving the equations of the continuous phase assuming no particles are present. Particles are then tracked through the continuous phase and particle equations are solved for particle velocity, temperature and mass, using the continuous phase parameters already calculated. The particle source terms are then calculated and the continuous phase equations solved again. This sequence is repeated until satisfactory convergence is attained (fig. 1).

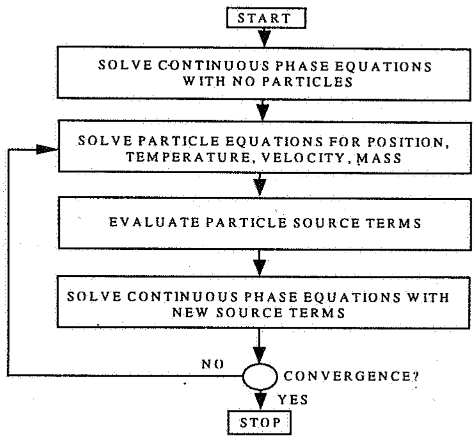

Figure 1. Flow chart for particle transport model computational sequence.

2.3 Continuous Phase Governing Equations

The code solves the following conservation equations for mass, momentum and energy (enthalpy) in the continuous (air) phase:

$$div(\rho U\phi) - div\left(\Gamma_\phi grad\phi\right) = S_\phi + S_\phi^P \tag{1}$$

Table 1. Terms in the governing equations when using an eddy viscosity turbulence model.

Conservation equation	ϕ	Γ_ϕ	S_ϕ
Mass	1	0	0
Momentum	U	$\mu+\mu_T$	$-gradp_0 + div(\mu_{eff} gradU) + \rho g$
Enthalpy	H	$\frac{\lambda}{C_P}+\frac{\mu_T}{\sigma_H}$	0
Mass Fraction of water vapour	Y_{wv}	Γ_{wv}	0

Note that in equation (1) there are two source terms, S_ϕ is the continuous phase source term and S_ϕ^P is the source term due to the particles. The continuous phase source terms and diffusion coefficients are given in table 1 for the (arbitrary) variable ϕ. All transient terms have been omitted since the work sought steady-state solutions.

p_0 is a 'modified' pressure given by

$$p_0 = p + \frac{2}{3}\rho k + \left(\frac{2}{3}\mu_{eff} - \zeta\right) gradU - \rho_{Bref}\mathbf{g}.\mathbf{x} \tag{2}$$

The turbulence model used in this work was the standard k-ε model [7].

2.4 The Particle Equations:

Momentum Equations

The equations for the rate of change of velocity of the particles come directly from Newton's second law:

$$m\frac{dU}{dt} = \underbrace{\frac{1}{8}\pi d^2\rho C_D|v_r|v_r}_{\text{Drag force}} - \underbrace{\frac{1}{4}\pi d^3\nabla P}_{\text{Pressure gradient}} + \underbrace{\frac{1}{6}\pi d^3(\rho_p - \rho)\mathbf{g}}_{\text{Buoyancy force}} - \underbrace{\frac{1}{6}\pi d^3\rho\frac{dU}{dt}}_{\text{Added mass force}} \tag{3}$$

where the drag factor C_D is given by : $C_D = 24\frac{(1+0.15\mathrm{Re}^{0.687})}{\mathrm{Re}}$

and Re is the particle Reynolds number: $\mathrm{Re} = \frac{\rho|v_r|d}{\mu}$

The Heat Transfer Equations

The particle rate of change of temperature is governed by two physical processes, convective heat transfer (Q_C) and latent heat transfer (Q_M) associated with mass transfer, where:

$$Q_C = \pi d\lambda \mathrm{Nu}\ (T_G\text{-}T_P) \qquad (4)$$

and

$$Q_M = \frac{dm}{dt}h_{fg} \qquad (5)$$

where the Nusselt number is given by $\mathrm{Nu} = 2 + 0.6\,\mathrm{Re}^{1/2}\,\mathrm{Pr}^{1/3}$

The total heat transfer is thus given by

$$mC_p\frac{dT}{dt} = Q_C + Q_M \qquad (6)$$

Mass Transfer Equations

Particle mass transfer is modelled using the 'Spray Drier Model' [5]. This controls the amount of mass transfer depending on whether a particle is above or below the 'boiling point'. A particle is said to be 'boiling' if the saturation vapour pressure at a given temperature, p_{sat}, is greater than the gaseous vapour pressure, where

$$p_{sat} = \exp\left[A - \frac{B}{T+C}\right] \qquad (7)$$

A, B and C are constants and their values for water are 23.196, 3816.44 and -46.13 respectively.

When the particle is below the boiling point, mass transfer is given by

$$\frac{dm}{dt} = \pi d D \mathrm{Sh}\frac{W_C}{W_G}\log\left(\frac{1-X}{1-X_G}\right) \qquad (8)$$

and when it is above the boiling point by

$$\frac{dm}{dt} = -\frac{Q_C}{h_{fg}}. \qquad (9)$$

Boundary Conditions

Three types of boundary condition were used in this investigation: WALL boundaries, PRESSURE boundaries, and SYMMETRY PLANE boundaries.

WALL boundary conditions are placed at fluid-solid interfaces and enable the specification of velocities (normally zero), heat fluxes, and temperatures. Conventional wall functions [6] are imposed at WALL boundaries.

Fluid may flow into or out of the domain across a PRESSURE boundary. If fluid flows *into* the domain, Neumann conditions (i.e. zero normal gradient) are imposed on velocity and turbulence quantities, and values assigned directly to pressure and temperature (Dirichlet conditions). When fluid flows *out* of the domain across a PRESSURE boundary, Dirichlet conditions are imposed on pressure, and Neumann conditions on all other variables.

At SYMMETRY PLANE boundaries, all variables are set to be mathematically symmetric, except the component of velocity normal to the boundary which is *anti*-symmetric. In 2D axi-symmetric flows a SYMMETRY *AXIS* is imposed at $r = 0$ at which the azimuthal (swirl) component of velocity is anti-symmetric.

3. The Problem being modelled

A cylindrical PDEC tower of 8.0m height and 1.0m diameter was modelled with adiabatic walls. Water droplets (diameter of $30\,\mu m$) were injected at 0.25m from the top of the tower with a temperature of 24°C and a volume flow rate of 6 l/h (speed = 30m/s). External conditions were zero wind, air temperature of 40°C, and relative humidity of 26.2% (water content of 12g water per kg dry air). These conditions are typical of those experienced in southern-European regions during the summer season.

4. CFD Representation of the Problem

In CFX-F3D two dimensional problems are defined by specifying a slice of the geometry which is one cell thick in the direction normal to the plane of the slice and imposing SYMMETRY PLANE boundaries on the two faces parallel to the slice. Consequently the PDEC tower was represented using a one radian slice with SYMMETRY PLANE boundaries set at the $\theta = 0$ and $\theta = 1$ rad faces. A PRESSURE boundary was placed at some (finite) distance from the tower to represent the exterior domain (figure 2). The external air conditions specified in section 3 are imposed at the PRESSURE boundaries.

The slice was divided into 20 cells in the radial direction and 120 cells in the longitudinal direction.

Water injection was represented using twenty vertically downward particle trajectories. Particles were injected from the centre of each cell along the tower width at 0.25m below the top of the tower with a mass flow rate directly proportional to the starting cell area. This ensured that the water was distributed evenly.

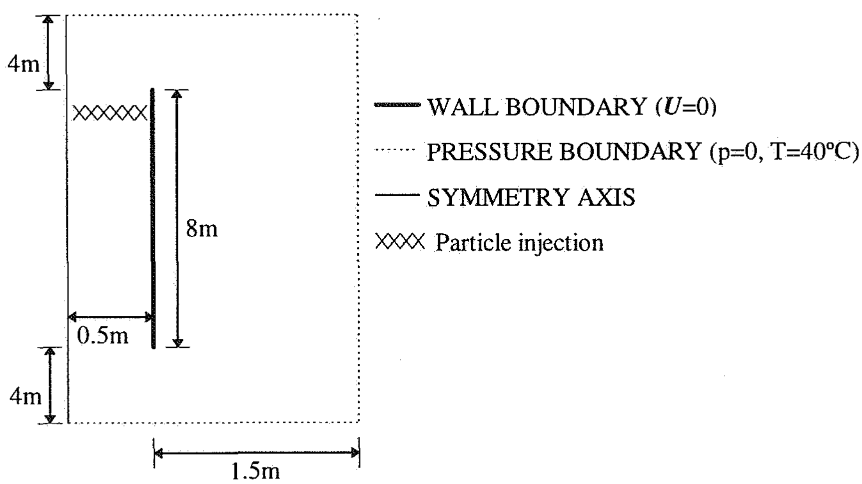

Figure 2. Geometry and boundary conditions used in the simulations.

5. Results and discussion

Injection of the water particles induces a downdraught shown in figure 3. The resulting particle tracks (fig. 4) are of uniform length and direction except in the vicinity of the wall where lower air speeds (due to wall friction effects) result in shorter tracks and small outward radial velocity component. Figures 5 and 6 illustrate the cooling effect produced by the particles. As expected, no further cooling occurs once the particles have evaporated.

The CFD results have been compared with those predicted by the one dimensional 'tower model' described in [2], [3] and [4]. In that model the flow variables are solved along the tower length. The tower was discretised into 100 elements and the flow variables calculated in each. In order to compare the CFD results with the tower model results, averaged values of the flow variables predicted by the CFD code were calculated at each height corresponding to the elements defined in the tower model.

The discrepancies between the CFD predictions and those of the 'tower model' suggest that the CFD code predicts a lower flow rate through the tower (fig. 7). This is thought to be due to 2D effects, in particular, turbulence, that are not present in the 'tower model'. As a result, in the CFD model, energy is given up in the production of turbulence and so less energy is available to bring about the mean flow parameters, thus yielding a lower flow rate. A reduced flow rate means there is less dry air available per unit mass of moisture and this yields a higher mass fraction of water (fig. 5) and cooler air which is also reflected in the CFD results (fig. 6). Another contributing factor to the higher flow rate in the 'tower' model is the assumption of zero pressure loss at the inlet.

The difference in the velocities between the CFD and 'tower model' at the top of the tower (fig. 7) is thought to be due to the momentum transfer between the particles and the air which is neglected in the 'tower' model. In the CFD code, momentum is transferred from the relatively fast moving water particles to the surrounding air. This causes acceleration of the air between the top of the tower and the injection point with a pressure drop in the same region (fig. 8). In the 'tower' model, particles are assumed to take the velocity of the surrounding air immediately after injection. Consequently there is no upstream acceleration, just a constant velocity equal to that at the tower inlet which ensures the mass entering the tower is equal to that leaving.

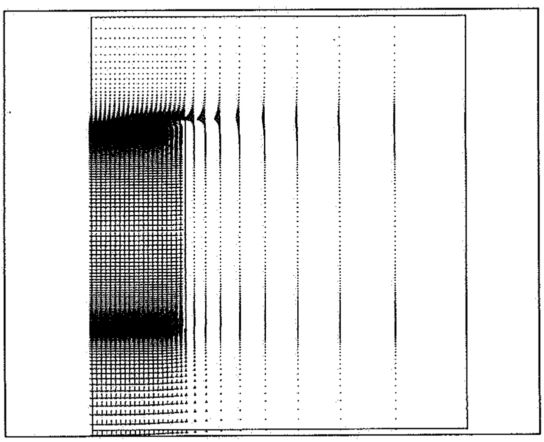

Figure 3. Flow pattern predicted by the CFD model.

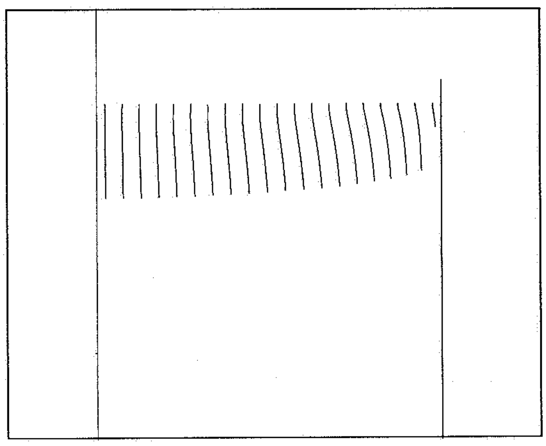

Figure 4. Particle tracks predicted by the CFD model (longest particle trajectory is 1.08m).

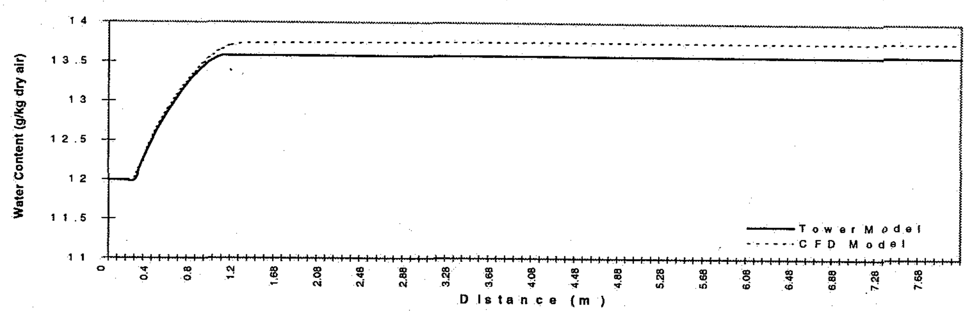

Figure 5. Water content along the tower length.

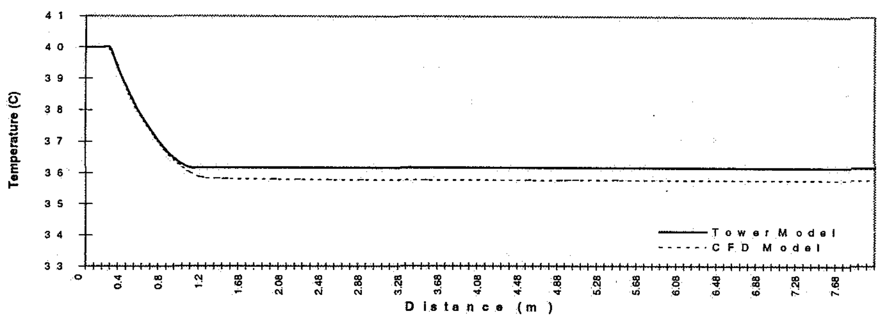

Figure 6. Air temperature along the tower length.

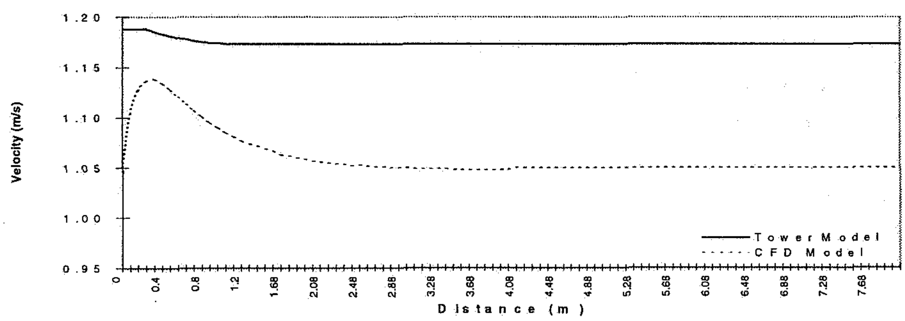

Figure 7. Vertical velocity magnitude of the flow along the tower length (0 = top of tower).

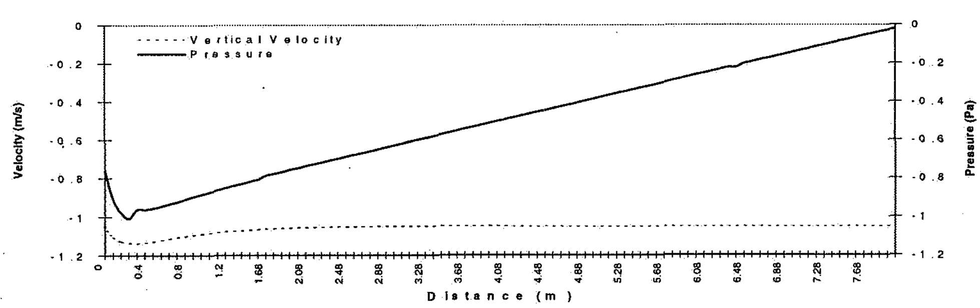

Figure 8. Vertical velocity and modified pressure along the tower length (CFD model).

6. Conclusions and Further Work

Predictions have been produced for a Passive Downdraught Evaporative Cooling (PDEC) system using CFD. In order to quantify the accuracy of the results, comparisons have been made with a one dimensional 'tower model'. The results compare favourably giving increased confidence in the CFD predictions. Some differences are explained and reasons for these are suggested.

It is now the intention to progress to a more accurate CFD model of the individual micronisers that are used for injecting the very fine mists of water. The work will identify optimum modelling techniques such as the number of particle trajectories required to accurately represent a single microniser and how to model size distribution of particles. Various numerical parameters used for obtaining convergence will also be investigated. It is then the intention to model a full (3D) PDEC system with wind catcher devices and delivery of cooled air into occupied spaces. The predictions will be compared with results from experiments currently under way in a test building at the Conphoebus Institute in Catania, Sicily.

7. Acknowledgment

The authors wish to thank Prof. E. Rodríguez for his comments and advice in the final stages of writing this paper.

8. References

1. Bowman, N., et al.
   "Application of Passive Downdraught Evaporative Cooling (PDEC) to Non-Domestic Buildings". Renewable Energy, Vol. 10., No. 2/3, 1997, pp.191-196.
2. Rodríguez, E. A., Alvarez, S., and Martin, R.
   "Direct Air Cooling From Water Drop Evaporation"
   Architecture and Urban Spaces, PLEA '91, 1991, pp.619-624.
3. Rodríguez, E. A., Alvarez, S., and Martin, R.
   "Water Drops As a Natural Cooling Resource. Physical Principles"
   Architecture and Urban Spaces, PLEA '91, 1991, pp. 499-504.
4. Alvarez, S., Rodríguez, E. A, and Molina, J. L.
   "The Avenue of Europe at EXPO '92: Application of Cool Towers"
   Architecture and Urban Spaces, PLEA '91, 1991, pp.195-201.
5. Computational Fluid Dynamics Services
   CFX 4.1 Flow Solver User Guide,1995 (AEA Harwell).
6. Versteeg H. K. and Malalasekera W.
   "An Introduction to Computational Fluid Dynamics - the Finite Volume Method"
   (1995) (Longman Group Ltd.)
7. Launder, B. E. and Spalding, D. B.
   "The numerical computation of turbulent flows"
   Computer Methods in Applied Mechanics and Engineering volume 3, 1974, pp269-289.