National Conference with International Participation
ENGINEERING MECHANICS 2001
Svratka, Czech Republic, May 14 – 17, 2001

NUMERICAL SIMULATION OF WET AIR FLOW WITH PHASE TRANSITION

I. Zúñiga-González*, F. Maršik†

Summary: *Two-phase flow in a partially closed nonadiabatic axisymmetric vertical channel is simulated by pressure relaxation technique. The lower end of the channel (cooling tower) is located in a certain inlet height from the flat plate (ground). The injection of cooling water caused the nonadiabatic character of the flow. The finite volume method is implemented assuming a laminar and turbulent viscous incompressible flow with phase transition in a staggered grid arrangement. The main aspects analysed were: the stability of the pressure-velocity coupling, the dependence on the grid size variation and on the variation of the under-relaxation parameters, and the droplet radius variation. The effects of the inlet height and of the channel geometry (channel is taken as a rotational hyperboloid) on the flow velocity and on the wet air transport can be discussed.*

Introduction

The methodology to analyse two-phase flows is well based on that already tried for single-phase flows. Basically three main types of assumptions have been made: (a) The 'homogeneous' flow model, considered the simplest one, in this model the two-phase flow is assumed to be a single-phase flow having pseudo-properties arrived at by suitably weighting the properties of the indiviual phases. (b) The 'separated' flow model in which the two phases of the flow are artifically segregated, then two sets of equations for each phase can be written, one for each phase, or alternatively the equations can be combined. In both cases information should be provided about the transversal area (in a channel flow) occupied by each phase (or alternatively, about the velocities of each phase) and about the frictional interaction betwen the phases. This information is then inserted into the basic equations, either from separated empirical relationships in which the void fraction and the wall shear stress are related to the primary variables, or on the basis of simplified model flows. (c) The 'flow pattern' model for which according to some previously prescribed geometry the two phases of the flow are arranged. Then the basic equations are solved within the correspondent idealised description.

Formulation

The governing equations for the flow of interest are described by the conservation laws for mass, momentun and energy. The following set of equations are based on the two-dimensional, laminar, axisymmetric, incompressible Navier-Stokes equations for a viscous Newtonian fluid:

$$\frac{\partial U}{\partial t} + \frac{1}{r}\frac{\partial (rE)}{\partial r} + \frac{\partial F}{\partial z} = H \tag{1}$$

*Ing.Israel Zúñiga-González, Ústav termomechaniky AVČR, Dolejškova 5, 182 00 Praha 8; tel. +420.2.6605.3392, e–mail: zuniga@it.cas.cz

†Prof.František Maršik, Dr.Sc., Ústav termomechaniky AVČR, Dolejškova 5, 182 00 Praha 8; tel. +420.2.8584695

where the independent variables are specified in U, the fluxes in E and F respectively:

$$U = \begin{pmatrix} \rho_a \\ \rho_v \\ \rho v_r \\ \rho v_z \\ \rho e \end{pmatrix}, \quad E = \begin{pmatrix} \rho_a v_r \\ \rho_v v_r \\ \rho v_r^2 + p \\ \rho v_r v_z \\ (\rho e + p) v_r \end{pmatrix}, \quad F = \begin{pmatrix} \rho_a v_z \\ \rho_v v_z \\ \rho v_z v_r \\ \rho v_z^2 + p \\ (\rho e + p) v_z \end{pmatrix} \tag{2}$$

and the sources in Q,

$$Q = \begin{pmatrix} 0 \\ w \\ \frac{p}{r} + \mu \left(\Delta v_r - \frac{v_r}{r^2} \right) \\ \mu \Delta v_z - \rho g \\ h_{ev} w \end{pmatrix} \tag{3}$$

where $\Delta \Phi = \frac{\partial^2 \Phi}{\partial r^2} + \frac{1}{r} \frac{\partial \Phi}{\partial r} + \frac{\partial^2 \Phi}{\partial z^2}$, with the following material relations:

$$D = 1.732 \times 10^{-9} T^{1.685} \tag{4}$$

$$h_{ev} = 3.16 \times 10^6 - 2.418 \times 10^3 T \tag{5}$$

$$\rho = \rho_a + \rho_v \tag{6}$$

$$\rho e = \rho_a c_{va} (T - T_0) + \rho_v c_{vv} (T - T_0) + \rho \frac{v^2}{2} \tag{7}$$

$$p = p_a + p_v = \left(\frac{\rho_a}{M_a} + \frac{\rho_v}{M_v} \right) RT \tag{8}$$

$$p_{vs} = exp \left(77.3491296 - \frac{7235.424651}{T} - 8.2 log T + 0.0057113 T \right) \tag{9}$$

$$M = \frac{4}{3} \pi \rho_e r_d^3 n_d \tag{10}$$

$$w = 4 \pi n_d \rho r_d D \left(\frac{p_{vs} - p_v}{p} \right) \tag{11}$$

Numerical method

The numerical simulation of the flow problem considered is based on the solution of the equations describing the conservation of mass, momentum and energy in a control volume. The volume considered is divided into a large number of small cells forming a grid. A FVM based on the pressure correction procedures devised by Patankar [3] and others, and the SIMPLE scheme (Semi-Implicit Method for Pressure-Linked Equations) are employed in the current simulation. The main feature of the FVM is that it deals with a set of discretised equations in a conservational form; furthermore, it employs an iterative solution strategy which results in a modest requirement in memory usage. The FVM is very popular in solving Newtonian flow problems at moderate to high Reynolds numbers. All of the governing equations can be written in the form of a general transport equation (summation convention is assumed):

$$\frac{\partial}{\partial t} (\Pi \Phi) + \frac{\partial}{h_j \partial x_j} (\Pi u_j \Phi) = \frac{\partial}{h_j \partial x_j} (\Gamma \frac{\partial \Phi}{h_j \partial x_j}) + S_\Phi, \tag{12}$$

where Φ represents either the velocity components or the volume fraction. When equation (12) $\Pi = 1$, $\Gamma = 0$ and $\Phi = \rho_a$ the equation of balance of air is recovered, when $\Pi = \rho$, $\Gamma = \eta$, and $\Phi = u_i$ one has the momentum equation in the i-direction, etc.

NOMENCLATURE

c_{va}	specific heat of dry air = 719.63 $[J \cdot kg^{-1} \cdot K^{-1}]$
c_{vv}	specific heat of steam = 1460 $[J \cdot kg^{-1} \cdot K^{-1}]$
D	diffusion coefficient $[m^2 \cdot s^{-1}]$
e	internal energy $[J \cdot kg^{-1}]$
h_{ev}	evaporation heat $[J \cdot kg^{-1}]$
p_{vs}	saturation pressure of water $[Pa]$
M	amount of cooled water in $1m^3$ $[kg \cdot m^{-3}]$
M_a	molecular mass of dry air = 28.964 $[kg \cdot kmol^{-1}]$
M_v	molecular mass of water vapor = 18.015 $[kg \cdot kmol^{-1}]$
n_d	droplet density $[m^{-3}]$
p	pressure $[Pa]$
r_d	droplet radius $[m]$
R	universal gas constant = 8314 $[J \cdot kg^{-1} \cdot K^{-1}]$
ρ	total density $[kg \cdot m^{-3}]$
ρ_a	density of dry air $[kg \cdot m^{-3}]$
ρ_b	density of water vapor $[kg \cdot m^{-3}]$
ρ_e	density of water = 10^3 $[kg \cdot m^{-3}]$
T_0	reference temperature = 273.15 $[K]$
T	temperature $[K]$
v_r	velocity in the radial direction $[m \cdot s^{-1}]$
v_z	velocity in the axial direction $[m \cdot s^{-1}]$
w	evaporation rate $[kg \cdot m^{-3} \cdot s^{-1}]$

Integrating the general transport equation over a finite volume V with bounding surface A, and applying the divergence theorem whenever needed, we get

$$\int_V \frac{\partial}{\partial t}(\Pi\Phi)dV + \int_A \Pi u_j \Phi n_j dA = \int_A \Gamma \frac{\partial \Phi}{\partial n} dA + \int_V S_\Phi dV. \tag{13}$$

The computational domain is now discretised into finite volumes on which the field variables are assumed to be piecewise constant. The integrations can now be performed numerically, and the discretised equations can be written in the form

$$a_P^t \Phi_P^t = \sum_{nb} a_{nb}^t \Phi_{nb}^t + a_P^0 \Phi_P^{t-\Delta t} + b^t, \tag{14}$$

where the subscript P represents the current node, the subscript nb represents a neighbouring node to P, the superscript t represents the current time, Δt is the time step, the coefficient a_P^t, a_{nb}^t and a_P^0 are found from the grid geometry and the current kinematics. To compute the gradient of any quantity, say ϕ, in a finite volume cell ΔV, it is taken to be constant, equal to its average value in the cell. Thus

$$\nabla\phi = \frac{1}{\Delta V}\int_{\Delta V} \nabla\phi dV = \frac{1}{\Delta V}\int_{\Delta S} \phi \mathbf{n} dS, \tag{15}$$

where ΔS is the bounding surface to the cell, and $\mathbf{n}$ is the outward unit normal vector on ΔS. The surface integral (line integral in two dimensions) is evaluated using interpolated values of ϕ between adjacent cells sharing the same boundary.

Numerical analysis

A cooling tower is a structure designed for the evaporative cooling of water by direct contact with air. The geometry of the cooling tower considered in this analysis (Figure 1), is described by using a general second-degree equation for its meridian as follows:

$$Az^2 + 2Brz^2 + CrR^2 + 2Dz + 2Er + F = 0 \tag{16}$$

in which r is the horizontal radius and z is the vertical coordinate along the axis of revolution. Two different equations are used: one to describe the meridian above the throat and the other to describe the meridian below the throat, with continuity of slope and curvature at the throat level (Table 1).

The grid system in the calculating region consists of 57 $\times$ 30 grid points in the radial and axial directions. A staggered mesh system is employed in the differential grid method in which the velocity (u_r, u_z) is defined at the cell edge and the pressure p, the density ρ and the internal energy e are at the center [4].

Boundary conditions

The boundary conditions corresponding to the case analysed in the geometry described by the cooling tower shown in Figure 1 are at the inlet boundary:

$$\frac{\partial v_r}{\partial r} = 0, \frac{\partial v_z}{\partial z} = 0.$$

At the outlet boundary:

$$v_r = 0, \frac{\partial v_z}{\partial z} = 0.$$

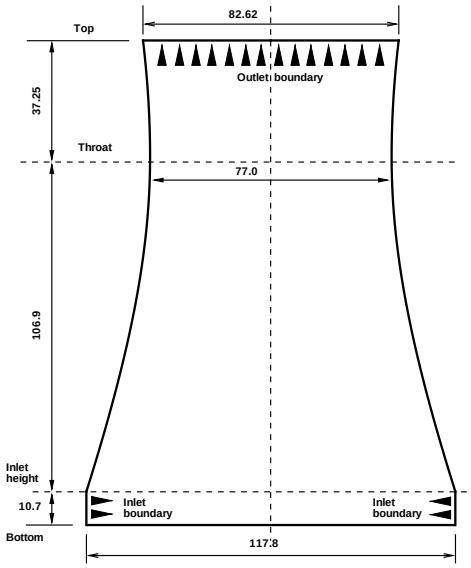

Figure 1: Geometry of cooling tower $[m]$.

Coeff.	Above throat level	Below throat level
A	-0.10667	-0.14350
B	-0.2755×10^{-5}	0.3087×10^{-5}
C	0.825	0.825
D	0.3108×10^{-3}	-0.2687×10^{-3}
E	-0.002619	-0.7118×10^{-3}
F	-13224.397	-13224.83

Table 1: Coefficients of equation (16).

Results and discussion

The numerical simulation was realised with the following parameters: boundary conditions, $v_r = -8.0$, $v_z = 0.0$; initial conditions: $v_z = 6.0$, $v_r = 0.0$. Underrelaxations factors for the

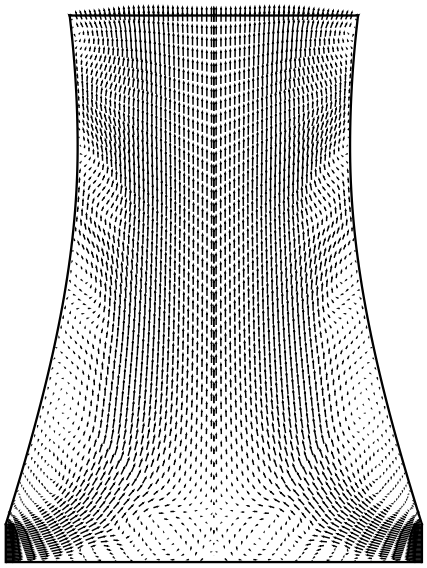

Figure 2: Velocity field

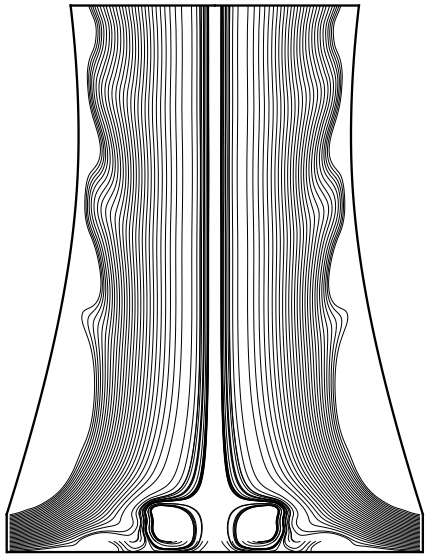

Figure 3: Streamlines

SIMPLE algorithm: $\beta_1 = 0.4$, $\beta_2 = 0.4$. The results can be appreciated in Figure 2 and Figure 3, which show the steady solution for the velocity field and the velocity streamlines respectively. Figure 4 shows the isobar plot of the model considered, all of them calculted on a mesh of 57×30 grid points. The vector velocity field shows a sudden expansion unidirectional to the outlet boundary, which can be better appreciated in Figure 3. The expectation of a pressure gradient near the center bottom is due to the pattern of the adiabatic boundary characterised by a notable increment in its value.

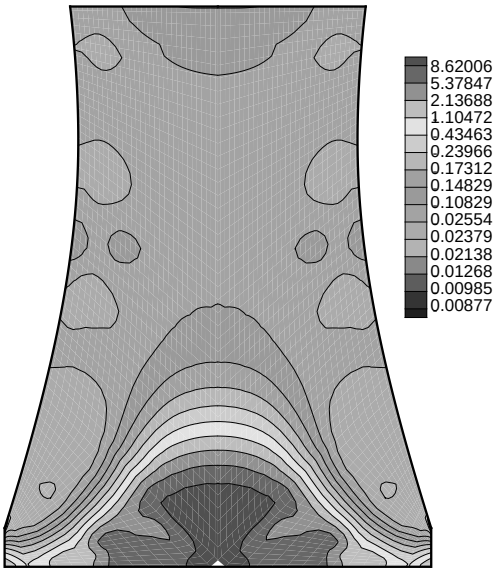

Figure 4: Isobars.

References

[1] Lun I., Calay R. K. and Holdo A. E. Modelling Two-Phase flows Using CFD. *Applied Energy*, **53**, 299–314, (1996).

[2] S.V. Patankar. *Numerical Heat Transfer and Fluid Flow.* Hemisphere Publishing Co., London, (1980).

[3] Ishii, M. *Thermo-Fluid Dynamic Theory of Two-Phase Flow.* Eyrolles, Paris, (1975).

[4] Uchiyama, T. Petrov-Galerkin finite element method for gas-liquid two-phase flow based on an incompressible two-fluid model. *Nuclear Engineering and Design*, **193**, 145–157, (1999.)