INT. J. NUM. METH. HEAT FLUID FLOW, VOL. 4, 269–280 (1994)

A MULTIPHASE FORMULATION FOR TWO PHASE FLOWS

E. DANIEL, R. SAUREL, M. LARINI AND J. C. LORAUD

IUSTI/SETT, Equipe Ecoulements Diphasiques et Réactifs, UA-CNRS 1168, Université de Provence, Centre de St-Jérôme, case 321, 13397 Marseille Cedex 20, France

ABSTRACT

This paper investigates the multi-phase behaviour of droplets injected into a nozzle at two separate wall locations. The physical features of the droplets (rate of mass, density and radius) at each injector location are identical. This system can be described by a two-phase Eulerian–Eulerian approach that yields classical systems of equations: three for the gaseous phase and three for the dispersed droplet phase.

An underlying assumption in the two phase model is that no interaction occurs between droplets. The numerical solution of the model (using the MacCormack scheme) indicates however that the opposite jets do interact to form one jet. This inconsistency is overcome in the current paper by associating the droplets from a given injection location with a separate phase and subsequently solving equations describing a multiphase system (here, three-phase system). Comparison of numerical predictions between the two-phase and the multiphase model shows significantly different results. In particular the multiphase model shows no jet interaction.

KEY WORDS Multiphase flow Droplets Eulerian–Eulerian approach MacCormack scheme

NOMENCLATURE

D diameter of droplets
H enthalpy
n number of droplets per unit volume
N total number of dispersed phases
P pressure
q heat transfer between gas and droplets
R radius of droplets
T temperature

Vectors

$\vec{F}$ drag force
$\vec{U}$ velocity
$\bar{I}$ identity square matrix

Subscripts

d droplet
k kth dispersed phase

Greek characters

α void fraction
ε total energy per unit mass
Γ mass transfer
μ gas viscosity
ρ density
τ dynamic relaxation time

INTRODUCTION

Gas-particle flows have been studied intensively during the last decade, involving more realistic phenomena. In aerospace and combustion applications, reactive or turbulent effects are often added into the formation of the equations governing these flows, to reach an accurate knowledge

0961–5539/94/030269–12$2.00

Received February 1993
Revised September 1993

of the flow field, in order to improve the design of liquid (or solid) rocket motors. One other centre of interest in such flows concern the multiphase flows due to injection of a dispersed phase into an initially one-phase flow[1,2]. This kind of problem could be found in the cooling of hot, high-speed jets or in instances related to nuclear reactor safety. Some recent numerical analyses regarding the influence of the injection of droplets on one-phase flow have been carried out in one-dimensional and two-dimensional geometries[3,4]. Numerical simulations have been hindered by the difficulty in treating the boundary between the one-phase and the two-phase flow. Essentially, the difficulties in these analyses are similar to those encountered when front tracking is performed. The model used is based on an Eulerian–Eulerian (or two-fluid) approach, where both the gaseous phase and the particle phase are treated as continua[5,6]. This approach is very useful when one is interested in solving the gas-particle flow on a scale which is large compared to the average spacing between particles. The main disadvantages are numerical instabilities and numerical diffusion. This last inconvenience could provide an explanation of the difficulty in calculating the propagation of a front of droplets. A Lagrangian treatment of the particle motion equations is an alternative method for tracking a front of droplets[7,8]. This technique is more accurate than the two-fluid one, but requires a large storage capacity to treat a large number of particles. In addition, the computational time is often much larger than that spent with a two-fluid model.

In this study, a two-fluid approach is used to simulate a double injection of droplets in a two-dimensional nozzle flow. The injectors are located in the divergent part on the lower and upper walls of the nozzle, resulting in a symmetrical configuration. Generally, the two-fluid model is regarded as a description of a multiphase flow with one gaseous phase and one dispersed phase. The dispersed phase is characterised by the radius of the droplets, their density and their velocity. This yields six differential vector equations: i.e. three for each phase. A more general description is obtained for multiphase flow by considering N dispersed phases: the N dispersed phases being identified by different values of a parameter, e.g., a first phase is constituted of droplets of radius r_1, and a second phase of droplets of radius r_2 ($r_1 \neq r_2$). In out case, the droplets injected from the two opposite injectors have initially the same features: same radius, same density and same rate of mass flow. Then a description of this dilute phase, by using the general equations (two-phase flows), is made and compared with a computation in which two dispersed phases are considered, in which each droplet stream is considered as a distinct dispersed phase. The results prove that the first approach is not compatible with the assumptions usually retained for such flow. One expects to find two jets of droplets crossing, but this only occurs when the second model is used. The concept of a droplet family is then established to give a more precise definition of a dispersed phase, including the history of the droplets.

MATHEMATICAL MODEL

The description of the two-phase flow produced by the injection of droplets in a gaseous nozzle flow uses an Eulerian–Eulerian model. Each phase is considered as a continuum. The main assumptions of the model are:

- The gas is inviscid and obeys the ideal gas law
- The dispersed phase is very dilute: any break-up, coalescence of droplets and interactions, e.g. droplet–droplet, droplet–wall, are ignored
- N dispersed phases could be injected into the flow ($N \geqslant 1$)

The last assumption means that there may be several dispersed phases in the flow. For example, two dispersed phases could differ by the density or by the average radius of the droplets. Then, a general system of unsteady equations is written for the gas and for the dispersed phases, in two-dimensional planar coordinates.

The gas phase equations are written in a conservative form[5,10]:

$$\frac{\partial}{\partial t}\rho\alpha + \vec{\nabla}\cdot\rho\alpha\vec{U} = \sum_{k=1}^{N}\Gamma_k$$

$$\frac{\partial}{\partial t}\rho\alpha\vec{U} + \vec{\nabla}\cdot\alpha(\rho\vec{U}\vec{U} + P\bar{I}_d) = \sum_{k=1}^{N}[\Gamma_k\vec{U}_{dk} - \vec{F}_{dk} + P\vec{\nabla}\alpha_{dk}] \tag{1}$$

$$\frac{\partial}{\partial t}\rho\alpha\varepsilon + \vec{\nabla}\cdot\alpha(\rho\varepsilon\vec{U} + P\bar{I}_d\cdot\vec{U}) = \sum_{k=1}^{N}\left[\Gamma_k\left(H_{dk} + \frac{|\vec{U}_{dk}|^2}{2}\right) - \vec{U}_{dk}\cdot\vec{F}_{dk} - q_k - P\frac{\partial\alpha_{dk}}{\partial t}\right]$$

The following equations apply to the kth dispersed phase:

$$\frac{\partial}{\partial t}\rho_{dk}(1-\alpha_{dk}) + \vec{\nabla}\cdot\rho_{dk}(1-\alpha_{dk})\vec{U}_{dk} = -\Gamma_k$$

$$\frac{\partial}{\partial t}\rho_{dk}(1-\alpha_{dk})\vec{U}_{dk} + \vec{\nabla}\cdot(1-\alpha_{dk})(\rho_{dk}\vec{U}_{dk}\vec{U}_{dk}) = -\Gamma_k\vec{U}_{dk} + \vec{F}_{dk} - (1-\alpha_{dk})\vec{\nabla}P \tag{2}$$

$$\frac{\partial}{\partial t}n_k + \vec{\nabla}\cdot n_k\vec{U}_{dk} = 0$$

The last equation expresses the fact that there is no break-up or coalescence of droplets in the flow. The energy equation for the dispersed phase is reduced to $T_d = T_{sat}(P)$: the droplet temperature is known and is equal to the saturation temperature which depends on the pressure.

The right-hand side of this system contains the interaction terms between the gas and dispersed phases. The gas phase equations have been written by using an superposition principle for the effects: the drag force acting on the gas is the sum of the drag forces of the N dispersed phases, and the same for all the others transfer terms. The different terms are:

- Γ mass transfer
- q heat transfer
- $\vec{F}$ drag force

The total system of equations is composed of three vector equations for the gas phase and $3*N$ vector equations for the N dispersed phases. In a majority of published works dealing with multiphase flows, N is equal to unity and the flow is a two-phase one. The system of equations is then reduced to six vector equations which can be obtained by letting $N = 1$ in the present model. There is no contradiction between the two models. The multiphase model is chosen here because it is the most general: it will be very useful to show the importance of the definition of a dispersed phase and it will be actually applied to the cases of $N = 1$ and $N = 2$. A discussion will be presented later in the paper regarding the definition of a dispersed phase and about the term 'family' which will be introduced as more convenient and illuminating.

The decomposition of this system along the x and y axes yields the following differential vector equation:

$$\mathbf{U}_t + \mathbf{F}_x + \mathbf{G}_y + \mathbf{H} = 0 \tag{3}$$

where

$$U = (\alpha\rho; \alpha\rho u_g; \alpha\rho\varepsilon_g; \cdots [(1-\alpha_{dk})\rho_{dk}; (1-\alpha_{dk})\rho_{dk}u_{dk}; n_{dk}]_{k=1,N})$$

The expression of the other vectors **F**, **G** and **H** is straightforward and will not be presented here.

The several void fractions appearing in the equations are now defined:

- α is the global void fraction appearing only in the gas equations. This quantity is obviously independent of the value of N and only related to the volume occupied by the gas and the

volume of thé dispersed phases. This quantity is given by the relation:

$$\alpha = \frac{\text{Volume occupied by gas}}{\text{Reference volume} = d\Omega}$$

(The reference volume $d\Omega$ is taken as a unity volume).

- α_{dk} is a pseudo-void fraction referring to the kth dispersed phase (a real void fraction of the kth phase must take into account the volume occupied by the gas and the other dispersed phases):
The volume occupied by this kth dispersed phase is $n_k \frac{4}{3}\pi R_k^3 * d\Omega$, then this void fraction α_{dk} is given by:

$$\alpha_{dk} = \frac{\text{Reference volume} - n_k \frac{4}{3}\pi R_k^3 d\Omega}{\text{Reference volume}} = 1 - n_k \frac{4}{3}\pi R_k^3$$

One could also define the presence of the kth phase by $\alpha_k = n_k \frac{4}{3}\pi R_k^3$, then the global void fraction α can be expressed by the general relation:

$$\alpha = 1 - \sum_{k=1}^{N} \alpha_k \tag{4}$$

Besides, the assumptions of dilute dispersed phases allows the assumption that terms $P\vec{\nabla}\alpha_k$ and $P\dfrac{\partial \alpha_k}{\partial t}$ negligible with regard to the transfer terms.

Expression of the transfer terms

The expression of the mass transfer is deduced from an energy balance around a droplet. The droplet temperature is that of saturation and the total heat coming from the gas is only utilised for vaporisation. Then the energy balance equation can be written as follows:

$$L_v \Gamma_k = q_k \quad \text{with } L_v \text{ the specific latent heat vaporisation}$$

A classical correlation with the Nusselt number is used to obtain the expression for the heat transfer. However phase change alters heat transfer. Yuen and Chen[11] who showed the diminution of heat transfer due to mass transfer proposed the application of a correction to the Nusselt number as follows:

$$\mathrm{Nu}_k = \frac{\mathrm{Nu}_k^*}{1 + B_k}$$

with $\mathrm{Nu}_k^* = 2 + 6.6\mathrm{Pr}_k^{1/3}\,\mathrm{Re}_k^{1/2} \cdot \mathrm{Nu}^*$ as a classical correlation for an inert particle. $B_k = \dfrac{H_g - H_{dk}}{L_{vk}}$, where H_k is the enthalpy per unit of mass of the phase k.

The drag force is taken as:

$$\vec{F}_k = \rho_{dk}(1 - \alpha_{dk})(\vec{U} - \vec{U}_{dk}) f_{dk}/\tau_k \qquad \text{with } \tau_k = D_k^2 \rho_{dk}/18\mu \qquad \text{and } f_{dk} = \frac{f_{dk}^*(\mathrm{Re}_k)}{1 + B_k}$$

The expression of f_d^* is issued from empirical correlations:

$$f_{dk}^* = \begin{cases} 1 + 0.15\mathrm{Re}_k^{0.687} & \mathrm{Re}_k < 1000 \\ 0.01833\mathrm{Re}_k & \mathrm{Re}_k > 1000 \end{cases}$$

The correction applied to the Nusselt number is extended to the drag coefficient as proposed by Chung and Olafsson[12]. This correction is not so strict as in the case of heat transfer but it is merely a way of taking into account alterations in the drag effects due to mass transfer.

The particle Reynolds number is given by:

$$\mathrm{Re}_k = \frac{\rho 2 R_{dk} |\vec{U}_{dk} - \vec{U}|}{\mu}$$

To evaluate these exchange terms, the diameter of the droplets at every time step and every location in the grid must be known. Because of the mass transfer, the size of the droplets changes and the new calculated diameter is given by the following relation:

$$D_k = 2\left[\frac{3(1-\alpha_{dk})}{4\pi n_k}\right]^{1/3}$$

The number of droplets per unit volume, n_k, is obtained directly from the system of equations while the void fraction is deduced from the relation $1 - \alpha_{dk} = [\rho_d(1-\alpha_{dk})]_{\text{computed}}/\rho_d$. This relation can be applied because the droplet density is constant.

Every thermophysical variables μ, γ are assumed to be constant, except the saturation temperature, the latent heat of vaporisation and the droplet enthalpy, which depend on the gas pressure. The liquid phase is in a state of equilibrium saturation.

NUMERICAL PROCEDURE

The two-step MacCormack explicit finite-difference scheme is applied to the partial differential equations in a computational square domain (ξ, η). The transformed equations take the following form:

$$\hat{\mathbf{U}}_t + \hat{\mathbf{F}}_\xi + \hat{\mathbf{G}}_\eta + \hat{\mathbf{H}} = 0 \tag{5}$$

with

$$\tilde{\mathbf{E}} = \mathbf{E}/\mathbf{D}$$
$$\tilde{\mathbf{F}} = (\xi_t \mathbf{E} + \xi_x \mathbf{F} + \xi_y \mathbf{G})/\mathbf{D}$$
$$\tilde{\mathbf{G}} = (\eta_t \mathbf{E} + \eta_x \mathbf{F} + \eta_y \mathbf{G})/\mathbf{D}$$
$$\tilde{\mathbf{H}} = \mathbf{H}/\mathbf{D}$$

where $\mathbf{D}$ is the Jacobian of the transformation.

Applying the algorithm for (5) yields a predictor-corrector scheme:

Predictor:

$$\tilde{\mathbf{U}}^* = \tilde{\mathbf{U}}^n - \Delta t\left(\tilde{\mathbf{H}}^n + \frac{\tilde{F}^n_{i-1,j} - \tilde{F}^n_{i,j}}{\Delta\xi} + \frac{\tilde{G}^n_{i,j-1} - \tilde{G}^n_{i,j}}{\Delta\eta}\right) \tag{6}$$

Corrector:

$$\tilde{\mathbf{U}}^{n+1}_j = \frac{1}{2}\left(\tilde{U}^n_j + \tilde{U}^*_j - \Delta t\left(\tilde{H}^*_j + \frac{\tilde{F}^*_{i+1,j} - \tilde{F}^*_{i,j}}{\Delta\xi} + \frac{\tilde{G}^*_{i,j+1} - \tilde{G}^*_{i,j}}{\Delta\eta}\right)\right) \tag{7}$$

The physical domain of integration is the entire nozzle. In this physical domain the lower wall of the nozzle can be represented by the function $RL(x)$ and the upper wall by $RU(x)$. This last relation is deduced from $RL(x)$ by symmetrical condition about the nozzle median plane. The coordinate transformation is given by:

$$\xi(x, y) = x/L$$
$$\eta(x, y) = \frac{y - RL(x)}{RU(x) - RL(x)}$$

where L is the nozzle length.

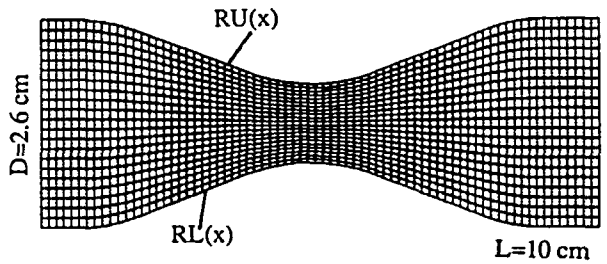

Figure 1 Grid for multiphase flow

Figure 1 shows the 75 * 22 computational grid used in this study. The length of the nozzle is 0.1 m, the convergent and divergent angles are equal to 15°. The geometrical features of the nozzle are quite simple: a solution of the multiphase flow in a complex geometry could mask some of fundamental effects we would point out.

Boundary conditions

Three kinds of boundary conditions in such a flow must be considered: inflow, outflow and wall.

At the wall

Computation of variables along the body surface is a difficult problem because if the gas phase is inviscid, then only one condition is generally assumed, the slip condition $\vec{V} \cdot \vec{n} = 0$. The treatment of boundary conditions is different for the gas phase and for the dispersed one.

Gas phase

The system of conservative equations (5) is solved using the MacCormack scheme. Spatial derivatives are approximated by an upwind second-order scheme[13]. For example, this technique applied to equations related to the upper wall leads to derivatives given by:

Predictor step:

$$\frac{\partial \tilde{\mathbf{G}}}{\partial \eta} = \frac{\tilde{\mathbf{G}}_{jm} - \tilde{\mathbf{G}}_{jm-1}}{\Delta \eta} \tag{8}$$

Corrector step:

$$\frac{\partial \tilde{\mathbf{G}}}{\partial \eta} = \frac{\tilde{G}_{jm} - 2\tilde{\mathbf{G}}_{jm-1} + \tilde{\mathbf{G}}_{jm-2}}{\Delta \eta} + \frac{\tilde{\mathbf{G}}^*_{jm} - \tilde{\mathbf{G}}^*_{jm-1}}{\Delta \eta} \tag{9}$$

(Then the scheme is also second-order accurate in space at the wall.)

After each stage the slip condition is computed as below:

$$v = -\eta_x/\eta_y u \tag{10}$$

Dispersed phase

The MacCormack scheme is also applied to compute the conservative dispersed phases variables at the wall. But if an upwind second order discretization is used, the solution cannot be reached because of instabilities. Then spatial derivatives are evaluated with a one-sided formula and relation (9) becomes:

$$\frac{\partial \tilde{\mathbf{G}}}{\partial \eta} = \frac{\tilde{\mathbf{G}}^*_{jm} - \tilde{\mathbf{G}}^*_{jm-1}}{\Delta \eta} \tag{11}$$

(Some other low-order methods can be applied to compute the droplet variables at the wall: e.g. linear extrapolation from interior points[5].)

A zero normal component of the velocity is also imposed for these phases.

Inflow and outflow boundary conditions

Gas phase

At the inlet, the flow is assumed to be adiabatic and isentropic. Compatibility equations along the direction $u_g - c$, where c is the sound velocity in the gas, are used to obtain the velocity component u, while component v is taken equal to zero.

$$\frac{dP}{dt} - \rho c \frac{du}{dt} = 0 \tag{12}$$

$$\left(\text{with } \frac{d}{dt} = \frac{\partial}{\partial t} + (u_g - c)\frac{\partial}{\partial x} \text{ for the equation along the direction } u_g - c.\right)$$

At the outlet cross section, if the outflow is subsonic, the ambient pressure is specified. Two compatibility equations along the directions u_g and $u_g + c$ are solved:

$$\frac{dP}{dt} - c^2 \frac{d\rho}{dt} = C(\gamma - 1) - c^2\left(\frac{\gamma - 1}{\gamma}\right)A \tag{13}$$

$$\frac{dP}{dt} - c\frac{d\rho}{dt} = C(\gamma - 1) + cB + \frac{c^2}{\gamma}A \tag{14}$$

The terms A, B, C represent respectively the source terms of the equation of mass, momentum and energy of the gas phase. They must appear in the equations to take into account the possible multiphase characteristics of the flow.

If the flow is supersonic, conservative variables are extrapolated.

Dispersed phase

At the inlet or at the location of injection, information such as velocity, diameter, density number is specified. At the outlet, the droplets are only allowed to exit the nozzle, then conservative variables are extrapolated.

RESULTS

The two-dimensional unsteady equations are solved in two different cases related to the number of dispersed phase: $N = 1$ and $N = 2$. In both cases, initial values of the gas phase variables are identical. A converged calculation of inviscid one-phase flow equations provides these data. The flow is supposed to be generated by a tank, with constant pressure (P_{tank}) and temperature (T_{tank}) even in a multiphase calculation. The initial condition is chosen to provide a subsonic flow: the Mach number at the throat vicinity is about 0.7.

The droplets are injected into the flow from the upper and the lower walls between the locations $x_1 = 7.1$ cm and $x_2 = 7.5$ cm ($\eta = 0$ and $\eta = 1$). Whatever the value of N, initial data for the dispersed phase are identical at the lower and upper injectors: same diameter, same rate of mass, same number of droplets per unit volume. The specification of this information at the injectors also represents boundary conditions which shall remain constant. At time $t = 0$ s, the difference between the cases $N = 1$ and $N = 2$ is not quantitative but qualitative. If $N = 1$ then computation consists in solving two-phase dynamics flow equations: one gaseous phase and one dispersed phase made up of droplets. If $N = 2$, the equations represent a multiphase flow: one gaseous phase and two dispersed phases: one is injected at the lower wall while the second at the upper wall. There is no quantitative difference between these two cases because of identical initial data. In *Table I* are summarised the main droplets values utilised in solving the multiphase flow equations.

Table 1 Multiphase flow data

Gas phase	Particle phase
$Cp = 1032.5\ \mathrm{J\,kg^{-1}\,K^{-1}}$	$D_0 = 50\ \mu\mathrm{m}$
$\gamma = 1.4$	$U_0 = 25\ \mathrm{m\,s^{-1}}$
$\mu = 1.3 \times 10^{-5}\ \mathrm{Pa{\cdot}s}$	$n_0 = 0.1 \times 10^{12}\ \mathrm{m^{-3}}$
$\mathrm{Pr} = 0.56$	Rate of mass $= 0.52\ \mathrm{kg\,s^{-1}}$
$P_{tank} = 4.5$ bars	
$T_{tank} = 900$ K	
Rate of mass $= 5.2\ \mathrm{kg\,s^{-1}}$	

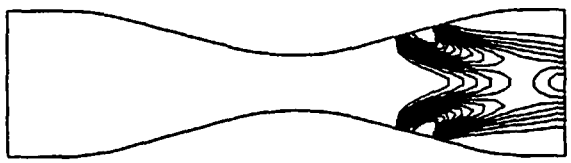

Figure 2 Solution of the system with $N = 1$. Normalised number of droplets per unit volume contours (n/n_0) ($n_{min} = 0$, $n_{max} = 1.065$, $\Delta n = 0.71 \times 10^{-1}$)

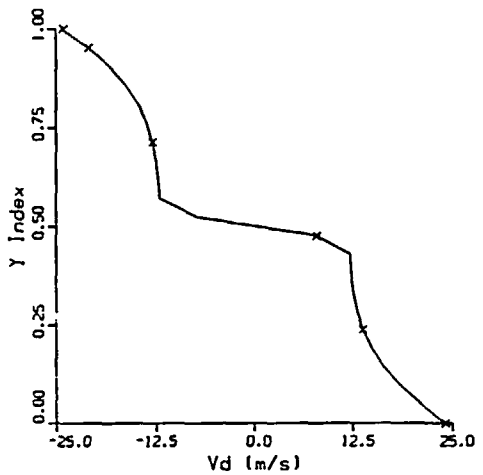

Figure 3 Solution of the system with $N = 1$. Cross section of the component v of the droplets velocity

1 Case N = *1*

As said previously, the equations for $N = 1$ correspond to a two-phase flow. In this case there is only one void fraction: $\alpha = \alpha_{d_1}$. The time step used to solve the system of equations is taken as 1/3 the CFL number. In the next analysis only the steady state is considered.

In *Figure 2* the iso-numbers of droplets per unit volume are plotted (these values have been normalised by the injected density number n_0). One can easily observe the positions of the injectors. One can notice that droplets do not fill the whole divergent: beyond the injectors there are two kinds of flow: a single-phase, near the wall and a two-phase region in the middle of the flow stream. Only one jet is observed at the exit from the nozzle. The two jets from the opposite injectors meet each other when they reach the axis of the nozzle and turn into a single jet.

One can also notice the discontinuity of the component v along the y axis of the droplets as shown in the *Figure 3*. On the nozzle axis, $v_d = 0\ \mathrm{ms^{-1}}$. This value can be explained by considering

fluxes in a computational cell on the axis ($j = J_{axis}$). Before droplets reach this position, the quantity $(1-\alpha)\rho_d v_d]^n$ is equal to zero. At the following time step, $(1-\alpha)\rho_d v_d]^{n+1}$ could change only with incoming fluxes of $(1-\alpha)\rho_d v_d$. Due to the centred approximation of the derivatives, the expression of fluxes at J_{axis} is evaluated with only the quantities at J_{axis+1} and J_{axis-1}. They have same value and opposite sign, then their sum is equal to zero, and this implies a resulting $(1-\alpha)\rho_d v_d]^{n+1} = 0$. This means that all the momentum along x and y is turning into momentum along x. If the profile of v_d is considered, it seems that the droplet flow on the side and the other side of the nozzle axis, ignored what happens on the axis. The value of v_d is developing due to drag action and yields the discontinuity. This behaviour is due to the parabolic nature of the system of equations of the dispersed phase: no information goes backwards to the injector, in other words the droplet front ignores what happens downstream. (See the Appendix regarding the nature of the system of equations.) This provides an explanation about the discontinuity of v_d.

The calculation of two opposite injections has been achieved that yields a two-phase flow with only one jet of droplets at the exit cross section. It is a trivial remark to say that the two jets have not crossed one another. This result is in contradiction with one assumption presented earlier: droplet–droplet interactions are neglected because the dispersed phase is very dilute. This assumption allows a crossing of the two jets of droplets, which does not occur in the results presented. We conclude that the model with $N = 1$ used before is not consistent with such initial and boundary conditions.

The multiphase flow produced by these two injections may be a two-phase flow in the common sense: a gaseous phase and a droplet phase. But strictly speaking, it is a multiphase flow, here a three-phase flow: one gaseous phase and two dispersed phases with identical thermo-dynamical and dynamical characteristics. The presence of the two different dispersed phases is explained by the fact that they must ignore each other. In order to distinguish the different cases we are now to define the term 'family' introduced previously in this paper.

2 Family of droplets

It is well known that if a particle is introduced into a gaseous flow, it is necessary to solve as many systems for the dispersed phase as there are different radii of the droplets. In other words, as many different particle families must be considered as there are different radii of droplets.

It is not the only case where the introduction of several families is necessary. An example is given by multiphase flows in which the dispersed phase is injected by N injectors ($N \geqslant 2$). Then, it is no longer possible to take into account the possible covering of a geometrical point of the flow by particles with different histories (here, particles coming from different injectors). So, one system of equations only is not able to describe such a flow.

It seems that, for such multiple-injections problems, it would be preferable to create several families, if:

- two particles may be at the same place in the same time
- one particle may be back at the same place at two different times

(This case should also be considered when the dispersed phase reflects on a wall, or more generally, changes the direction.)

Then the case $N = 2$, named multiphase flow, could be called 'injection of two families of droplets'.

3 Case N = 2

Now the results of the injection into the flow of two families of droplets are presented. The system integrated here is formed by 12 differential scalar equations: 4 for the gas, 4 for the first family ($k = 1$, injected from the lower wall), and 4 equations for the second family ($k = 2$, from

the upper wall). The time step used is equal to 1/3 the CFL number and only the steady state is analysed. Initial and boundary conditions are the same as those presented before.

In *Figure 4*, the iso-number curves of droplets per unit volume are plotted for the families 1 and 2. The evolution of this variable is absolutely symmetrical, because of the symmetrical initial and boundary conditions. In *Figure 5* the previous contours are superposed in order to show where the multiphase flow is exactly in the nozzle. Clearly, in this image, the crossing of the two jets can be easily observed.

The different treatment of the multiphase flow in these two cases ($N = 1$ and $N = 2$), alters the gas evolution. In *Figure 6* the isovelocity lines of the gas phase are drawn for $N = 1$ and $N = 2$. In both cases symmetrical isolines are observed, but differences may be seen between

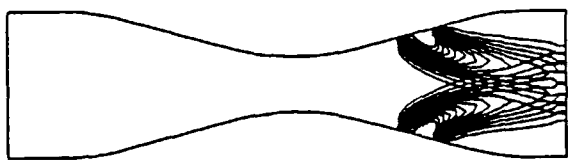

Figure 4 Solution of the system with $N = 2$. Normalised number of droplets per unit volume contours of family 1 (n_1/n_0), and family 2 (n_2/n_0). ($N_{min} = 0$, $N_{max} = 1.055$, $\Delta n = 0.70 \times 10^{-1}$)

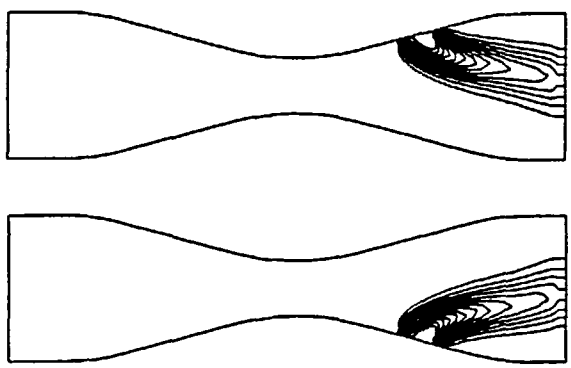

Figure 5 Solution of the system with $N = 2$. Superposition of the number of droplets per unit volume contours of families 1 and 2. ($N_{min} = 0$, $N_{max} = 1.055$, $\Delta n = 0.70 \times 10^{-1}$)

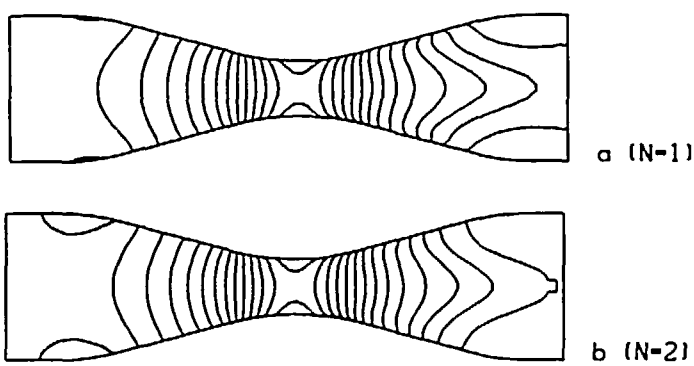

Figure 6 Gas velocity. (a) Solution of the system with $N = 1$ ($V_{min} = 82$, $V_{max} = 338$, $\Delta v = 17\ ms^{-1}$); (b) solution of the system with $N = 2$ ($V_{min} = 91$, $V_{max} = 349$, $\Delta v = 17.2\ ms^{-1}$)

these two figures. One can notice a difference about $10\,\mathrm{ms}^{-1}$ between the minimum and the maximum values of the velocity between these two cases. The gas flow is affected in different ways by the presence of droplets or, more precisely, by the presence of one or two families.

CONCLUSION

The numerical simulation of the flow of droplets through two injectors in a gaseous compressible flow has been achieved and several conclusions can be reached as a result of this study. We were not interested in the quantitative data: results are significant enough to be discussed without considering actual values. The main problem encountered in this study lies in the usual definition of a two-phase flow, which might be not representative of reality. In previous two-phase flow studies, one only considers a gaseous phase and a dispersed phase, the last phase being known when information such as diameter, velocity and density is given. In the case of multiple injections, it is shown that this definition is not general enough and could lead to false results. Here, the calculation should have shown two crossing droplets jets according to the assumptions made, and this phenomenon was not observed. So, a more accurate description of the dispersed phase is proposed, where history is taken into account.

The notion of family has been introduced, which yields a more complicated system of equations. The solution of the system has shown the expected crossed jets.

One can reasonably think that this is not the real phenomenon: when jet crossing is simulated the assumption of no droplet–droplet interaction may be no longer valid. But the results obtained seem to show that the numerical treatment with N families is more realistic than with only one family. The definition of a family may not be exhaustive. For example, this problem is also encountered when a family of droplets reaches a wall: droplets may stick or reflect but they do not belong any more to the same family as before hitting the wall.

APPENDIX

The nature of the system of equations for a dilute multiphase flow is now studied in the one-dimensional case. Then, every gradient terms of the right-hand side can be neglected. The system is written in one dimensional form: $\mathbf{U}_t + \mathbf{A}\mathbf{U}_x + \mathbf{H} = 0$ where $\mathbf{A}$ is the Jacobian matrix $\left(\mathbf{A} = \dfrac{\partial \mathbf{F}}{\partial \mathbf{U}}\right)$.

A change of variables is performed in order to facilitate the demonstration. The components of the vector $\mathbf{U}$ are now given by:

$$\mathbf{U} = (r_g = \alpha\rho;\ m_g = \alpha\rho u_g;\ E_g = \alpha\rho\varepsilon_g;\ \cdots [r_{dk} = (1-\alpha_{dk})\rho_{dk};\ m_{dk} = (1-\alpha_{dk})\rho_{dk}u_{dk};\ N_{dk} = n_{dk}]_{k=1,N})$$

There is no difficulty in finding that the Jacobian matrix takes the following form:

$$\mathbf{A} = \begin{bmatrix} [A_g] & [0] & [0] & [0] \\ [0] & [A_{d,1}] & [0] & [0] \\ [0] & [0] & \cdots & [0] \\ [0] & [0] & [0] & [A_{d,N}] \end{bmatrix}$$

A is a block diagonal matrix, with block $\mathbf{A}_g$ related to the gaseous phase and $\mathbf{A}_{dk}$ to the kth dispersed phase. The nature of the system is obtained by finding the eigenvalues of $\mathbf{A}$, given by the solution of $\mathbf{DET} = \det(\mathbf{A}^{\mathrm{T}} - \lambda I_d) = 0$. The diagonal form of $\mathbf{A}$ allows us to write:

$$\mathbf{DET} = \det(\mathbf{A}_g^{\mathrm{T}} - \lambda_g \bar{I}) * \prod_{k=1}^{k=N} \det(\mathbf{A}_{dk}^{\mathrm{T}} - \lambda_{dk}\bar{I})$$

The eigenvalues are:

$$\lambda_{g1} = m_g/r_g - c = u_g - c$$
$$\lambda_{g2} = m_g/r_g = u_g$$
$$\lambda_{g2} = m_g/r_g + c = u_g + c$$

with

$$c^2 = \gamma(\gamma - 1)\frac{E_g}{r_g} - \frac{m_g^2}{r_g^2}\frac{\gamma}{2}(\gamma - 1) = \gamma RT$$

and

$$\lambda_{dk1} = \lambda_{dk2} = \lambda_{dk3} = m_{dk}/r_{dk} = u_{dk} \qquad (k = 1, N)$$

The three eigenvalues of the kth dispersed phase are real and equal.

If each phase is considered then the gas equations are hyperbolic, and the equations for each dispersed phase equations are parabolic. The whole system is a degenerate hyperbolic one.

REFERENCES

1 Daniel, E., Loraud, J.-C. and Larini, M. Influence de l'injection de gouttes d'eau dans de la vapeur en écoulement, *Int. J. Heat and Mass Transfer*, **36**, no. 6, pp. 1619–1632 (1993)
2 Hu, C. K. and Yu, S. Z. Numerical calculation of gas-liquid two-phase flow in a dump combustor, *Proc. 7th Int. Conf. Num. Meth. Laminar and Turbulent Flow*, pp. 804–812, Stanford (1991)
3 Daniel, E., Larini, M. and Loraud, J.-C. Numerical modelling of single phase two-phase flow transition by injection of droplets, *Num. Meth. Laminar and Turbulent Flow*, **7**, 1701–1709 (1991)
4 Daniel, E., Larini, M., Loraud, J.-C. and Porterie, B. *AIAA Paper 92-2929*, Nashville, Ten. (1992)
5 Shih Chang, I. One and two-phase nozzle flows, *AIAA J.*, **18**, 1455–1461 (1980)
6 Shih Cheng, I. Three-dimensional two-phase supersonic nozzle flows, *AIAA J.*, **21** (5), 671–678 (1983)
7 Mehdi Golafshani and Hai-Tien Loh. Computation of two-phase viscous flow in solid rocket motors using a flux-split Eulerian-Lagrangian technique, *AIAA/ASME/ASEE 25th Joint Propulsion Conf.*, Monterey (1989)
8 Madabhushi, R. K., Sabnis, J. S., de Jong, F. J. and Gibeling, H. J. Navier-Stokes analysis of aft dome flow field in solid rocket motors with submerged nozzle, *AIAA-89-2780* (1989)
9 Sivier, S., Loth, E., Baum, J. and Lohner, R. Unstructured adaptive remeshing FEM for dusty shock flows, *AIAA Paper 92-3443*, Nashville, Ten. (1992)
10 Olim, M., Igra, O. and Ben-Dor, G. A general attenuation law of planar shock waves propagating into dusty gases, *Proc. 16th Int. Symp. Shock Tube*, 217–225 (1987)
11 Yuen, M. C. and Chen, L. W. Heat transfer measurements of evaporating liquid droplets, *Int. J. Heat and Mass Transfer*, **21**, 537–542 (1977)
12 Chung, J. N. and Olafsson, S. I. Two-phase droplet flow convective and radiative heat transfer, *Int. J. Heat and Mass Transfer*, **27**, 901–910 (1983)
13 Warming, R. F. and Beam, R. M. Upwind second-order schemes and applications in aerodynamic flows, *AIAA J.*, **14** (9) (1976)