Two-Phase Flow Computations using a Two-Fluid Model and Fluctuation Splitting Schemes

Eusebio Valero*, Mario Ricchiuto† and Gérard Degrez‡

von Kármán Institute for Fluid Dynamics (VKI)
Chaussée de Waterloo, 72
B-1640 Rhode-Saint-Genèse, Belgium
Tel: (32) 2 359 96 11. Fax: (32) 2 359 96 00.

Abstract

This work is devoted to the application of residual distribution schemes to two-dimensional two-phase problems. This numerical scheme is based on the fluctuation-splitting methodology. The solution is approximated by a continuous function, varying linearly over each element (triangle in $2D$, segment in $1D$) like in finite element methods. Then, the residual in each element is defined as the time variation of the system variables. Finally, making use of the hyperbolic properties of the system the residual is distributed to the nodes of the triangle. This scheme has been successfully applied to Euler and Navier-Stokes equations in three-dimensional non-structured grids where it has been proved to be second order accurate in steady solution. Now, it is applied to two-phase flows. The 2D-two-phase, one-pressure model has been chosen and Two different test cases has been considered. First, the water faucet problem has been chosen to check the validity of the method. Second, a fully inviscid two-phase flow around a NACA0012 airfoil is shown.

Introduction

THE numerical simulation of two-phase flows constitutes, even today, one of the most open and difficult problems in fluid mechanics. However, its study is of great interest in industry, since this kind of flows is encountered in a wide variety of engineering applications: power systems and energy conversion (internal combustion engines or jet engines), heat transfer processes (evaporators, boilers or condensers), industrial processes (distillation, chemical reactors), nuclear technology, transport systems (air-lifts, ejectors), environmental control (air conditioning, refrigerators), meteorology (clouds dynamics), biological system (blood flows) and so on.

The difficulty of two-phase flows lies in the mathematical model you have to manage. In principle, the local instantaneous conservation equations for each phase may be written down together the appropriate initial interfacial boundary conditions. The resulting moving boundary value problem is intractable except in the simplest cases. However, in the majority of industrial applications, the local instantaneous behavior of the different flow variables is not required, only averaged quantities appear to be sufficient. Therefore, one of the main approaches to two-phase modeling has been to average (in time, space, an ensemble, or in some combination of these) the local instantaneous conservation equations. Interesting works about this subject can be found in Ishii,[1] Drew,[2,3] Banerjee & Chan.[4] In the averaging process, a lot of interfacial information is lost, which must be supplied by means of auxiliary relations. The resulting equations, although they form a mathematical model much simpler than the original formulation, are still difficult to manage.

From a numerical point of view, although the original system of equations is hyperbolic, the resulting averaged system has complex characteristics (Stewart[5]). Therefore, it is ill posed as an initial value problem (Ransom & Hicks[6]). Real value characteristics are necessary for initial value problems to be well posed (Courant & Hilbert[7]). Many efforts has been devoted to rendering the characteristics real by adding first order differential terms to the momentum equation (terms that have been neglected during the averaging process), like the relative acceleration

*Ph.D. researcher at VKI, Department of Aeronautics and Aerospace. E-mail: valero@vki.ac.be.
†Rechearch Engineer at VKI, Department of Aeronautics and Aerospace. E-mail: ricchiut@vki.ac.be.
‡Professor at VKI, Department of Aeronautics and Aerospace. E-mail: degrez@vki.ac.be.

(Cortes et al.[9]). Other authors consider that the inclusion of source or viscous terms make the system stable (Arai[10]), although care must be taken in long wavelength. Pokharna et al.[11] consider that the viscosity of the numerical scheme can also produce the same stabilization effect.

This works is devoted to the application of residual distribution schemes to two- phase problems. This numerical scheme has been successfully applied to Euler and Navier-Stokes equations in three-dimensional non-structured grids (Paillere[12]). In the first section we introduce the two fluid-one pressure model. Such a model is studied in the following sections. First, the hyperbolycity of the model is analysed. The perturbation technique, described by Toumi,[13] is used in order to prove the existence of real eigenvalues. Second, the residual distribution schemes is introduced, and its application to two-phase flow is described. Lastly, two different cases has been checked. The first one, the faucet problem, which has been proposed by Ransom,[14] consists of a vertical water jet, contained within a cylindrical channel, that is accelerated under the action of gravity. Due to the existence of an analytical solution, this problem has been used by different authors (Ghidaglia et al.,[15] Tiselj & Petelin[16] among others). The second shows the capabilities of the approach used here to solve flows involving multidimensional physics. The two-phase flow over a NACA0012 airfoil is studied and its effect on the pressure distribution is analysed.

The two fluid-one pressure model

The six equations-one pressure model is analysed. This model is formed by two phasic mass equations and two phasic vector momentum equations. We do not deal with the two phasic energy equations at this point, alternatively we consider isentropic flow and incompressibility for the liquid phase. Also we do not consider phase change and bulk viscosity in each phase. With the above assumptions the system of equations for the liquid and vapor phases read,

- Mass conservation equations.

$$\frac{\partial(\alpha_k\rho_k)}{\partial t} + \nabla\cdot(\alpha_k\rho_k\vec{V}_k) = 0, \qquad (1)$$

- Momentum conservation equations.

$$\begin{array}{l}\frac{\partial(\alpha_k\rho_k\vec{V}_k)}{\partial t} + \nabla\cdot(\alpha_k\rho_k\vec{V}_k\otimes\vec{V}_k) \\ +\alpha_k\nabla p = \alpha_k\rho_k\vec{f}^{ext} + \vec{F}_k^{int},\end{array} \qquad (2)$$

field acting on the system, (usually gravity force), and $\vec{F}_k^{int}$ is due to momentum exchange at the interface. Conservation of momentum at the interfacial level implies

$$\sum_{k=g,l}\vec{F}_k^{int} = \vec{M}^{\sigma}, \qquad (3)$$

where $\vec{M}^{\sigma}$ is a mixture momentum source term.

- Energy equations.

$$s_k = s_k(p, h_k), \qquad (4)$$

where s_k is the phasic entropy and h_k is the phasic enthalpy, the hypothesis of constant entropy implies that the energy (enthalpy) field can be obtained from equation (4) once the pressure field has been solved. As a consequence of that, no energy equations need to be solved.

The variables appearing in the above equations have the following meanings (here we set $k = g, l$ for the vapor and liquid phases):

α_k volume fraction of k-phase,

ρ_k density of k-phase,

$\vec{V}_k$ velocity of k-phase,

p common pressure to the two-phases,

s_k entropy of k-phase.

Equations (1) and (2) do not represent a closed mathematical problem, so some closure relations must be specified in order to solve it. In the course of this work the following relations have been used:

- Gas state equation:

  The liquid is assumed incompressible, and the gas is supposed to be ideal and following an isentropic process: state:

$$\rho_l = constant, \quad p = \Gamma\rho_g^{\gamma}. \qquad (5)$$

- External body forces:

  Only gravity forces are considered here, so

$$\vec{f}^{ext} = \vec{g}. \qquad (6)$$

The interfacial momentum transfer is present in the equations in the term $\vec{F}_k^{int}$, and it contains both viscous and inviscid interactions at the interface. So it can be written:

$$\vec{F}_k^{int} = (\vec{F}_k^{int})^v + (\vec{F}_k^{int})^{inv}, \tag{7}$$

where v stands for viscous and inv for inviscid. The viscous part of $\vec{F}_k^{int}$ is the drag force between phases and must be modeled in some way. In general the following form for interfacial drag forces is found in literature:

$$\begin{array}{c}(\vec{F}_l^{int})^v = K_D \, \|\vec{V}_r\| \, \vec{V}_r, \\ (\vec{F}_g^{int})^v = -(\vec{F}_l^{int})^v, \\ K_D = \frac{3}{8} \, \frac{C_d}{R} \, \rho_{cont} \, \alpha_{disp}\end{array} \tag{8}$$

where C_d is a drag coefficient and R is an averaged bubble (or droplet) radius.

The inviscid interactions at the interface can be decomposed as follows:

$$(\vec{F}_k^{int})^{inv} = \vec{F}_k^{VM} + \vec{\Pi}_k + \vec{L}_k, \tag{9}$$

where

- $\vec{F}_k^{VM}$ is the virtual mass force acting on phase k;
- $\vec{\Pi}_k$ is the interface pressure action on phase k;
- $\vec{L}_k$ is the lift force acting on phase k.

Usually these actions are modeled with differential terms that influence the eigenstructure of the system. An extensive review of all the possible terms contained in $(\vec{F}_k^{int})^{inv}$ can be found in Kumbaro et al.[17] and a discussion on their conceptual derivation in Drew.[2] Here we will follow the work made in Cortes et al.[9] and consider only the interface pressure effects.

This term assumes a local pressure discontinuity at the interface and has the following form:

$$\vec{\Pi}_k = -(p - p_k^{int}) \nabla \alpha_k. \tag{10}$$

Several interface pressure correction terms exist in the literature and most of them model the pressure difference as a frictional term: $p - p_k^{int} \approx \|\vec{V}_r\|^2$. In this work, the Lahey[18] pressure correction has been assumed, although in this case the model does not satisfy global momentum conservation.

$$\begin{array}{ll} p - p_g^{int} & = 0, \\ p - p_l^{int} & = C_P \alpha_g \, \rho_l \, \|\vec{V}_r\|^2, \end{array} \tag{11}$$

The form of Eqs (11) is justified as follows: because of the small gas density, the gas phase cannot support large pressure gradients at the interface, thus volume changes (expansion/contraction) in the bubbles give a contribution to $(p - p_k^{int})$ only in the liquid region. This is especially true for bubbly flows where the gas occupies relatively small volumes.

Under these assumptions the equations can be rewritten as follows:

- Liquid phase equations:

$$\begin{array}{c} \frac{\partial(\alpha_l \rho_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l \vec{V}_l) = 0, \\ \frac{\partial(\alpha_l \rho_l \vec{V}_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l \vec{V}_l \otimes \vec{V}_l) + \alpha_l \nabla p = \\ \alpha_l \rho_l \vec{g} + K_D \|\vec{V}_r\| \vec{V}_r - C_P \alpha_g \rho_l \|\vec{V}_r\|^2 \nabla \alpha_l. \end{array} \tag{12}$$

- Gas phase equations:

$$\begin{array}{c} \frac{\partial(\alpha_g \rho_g)}{\partial t} + \nabla \cdot (\alpha_g \rho_g \vec{V}_g) = 0, \\ \frac{\partial(\alpha_g \rho_g \vec{V}_g)}{\partial t} + \nabla \cdot (\alpha_g \rho_g \vec{V}_g \otimes \vec{V}_g) + \alpha_g \nabla p = \\ \alpha_g \rho_g \vec{g} - K_D \|\vec{V}_r\| \vec{V}_r. \end{array} \tag{13}$$

Approximate eigenstructure of the system

In this section we will see that there is not analytical expressions for the eigenstructure of the system, so we must manage only approximate values of the eigenvalues and eigenvectors.

To see the order of magnitude of the different terms involved in the equation, non-dimensional variables are taken. Furthermore, we follow the density perturbation method developed by Cortes,[9] although, in our case, we take physical quantities for the whole system, not only for densities.

Non-dimensional equations

In that way, we now non-dimensionalize the motion equations by introducing reference scales for all variables,

$$\begin{array}{c} \tilde{x} = x/L, \quad \tilde{t} = t/t_r \quad \tilde{\rho}_l = \rho_l/\rho_{lr}, \\ \tilde{\rho}_g = \rho_g/\rho_{gr}, \quad \tilde{u}_g = u_g/v_r, \quad \tilde{u}_l = u_l/v_r, \quad \tilde{p} = p/p_r. \end{array}$$

Here, L represents a characteristic length scale of the problem. The density for liquid (ρ_{lr}) and gas (ρ_{gr}) are taken as representative thermodynamic

reference freestream velocity, and it is assumed the same for both liquid and gas, provided the slip ratio (difference between liquid and gas velocities) is small. The reference time (t_r) is taken as the characteristic residence time (L/v_r), and finally the reference pressure (p_r) is made equal to the dynamic pressure in the gas ($\rho_g v_r^2$), assuming that the pressure waves travel in the gas.

The resulting non-dimensional equations can be written in a matrix form as:

$$\frac{\partial \vec{U}}{\partial t} + \mathbf{A}\frac{\partial \vec{U}}{\partial x} + \mathbf{B}\frac{\partial \vec{U}}{\partial y} = \vec{S} \tag{14}$$

the $\mathbf{A}$ and $\mathbf{B}$ matrices are given below:

$$\mathbf{A} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ \alpha C_P\|\vec{V}_r\|^2 - u_l^2 + \frac{\epsilon(1-\alpha)\gamma p}{\alpha} & 2u_l & 0 & \frac{\epsilon(1-\alpha)\gamma p}{\alpha\rho_g} & 0 & 0 \\ -u_l v_l & v_l & u_l & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ \gamma p & 0 & 0 & \frac{\gamma p}{\rho_g} - u_g^2 & 2u_g & 0 \\ 0 & 0 & 0 & -u_g v_g & v_g & u_g \end{bmatrix} \tag{15}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 0 & 1 & 0 & 0 & 0 \\ -u_l v_l & v_l & u_l & 0 & 0 & 0 \\ \alpha C_P\|\vec{V}_r\|^2 - v_l^2 + \frac{\epsilon(1-\alpha)\gamma p}{\alpha} & 0 & 2v_l & \frac{\epsilon(1-\alpha)\gamma p}{\alpha\rho_g} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & -u_g v_g & v_g & u_g \\ \gamma p & 0 & 0 & \frac{\gamma p}{\rho_g} - v_g^2 & 0 & 2v_g \end{bmatrix}. \tag{16}$$

where the small parameter $\epsilon = \rho_{gr}/\rho_{lr}$ has been introduced.

And, the source term given by,

$$\vec{S} = \begin{bmatrix} 0 \\ (1-\alpha)\ (\vec{g})_x + \frac{K_D(\vec{U})}{\rho_l}\|\vec{V}_r\|u_r \\ (1-\alpha)\ (\vec{g})_y + \frac{K_D(\vec{U})}{\rho_l}\|\vec{V}_r\|v_r \\ 0 \\ \alpha\ \rho_g\ (\vec{g})_x - \frac{K_D(\vec{U})}{\epsilon\rho_l}\|\vec{V}_r\|u_r \\ \alpha\ \rho_g\ (\vec{g})_y - \frac{K_D(\vec{U})}{\epsilon\rho_l}\|\vec{V}_r\|v_r \end{bmatrix}. \tag{17}$$

Hiperbolicity of the system

The system is hyperbolic if the matrix $\mathbf{C}=\mathbf{A}n_x+\mathbf{B}n_y$ has real eigenvalues $\forall\vec{n}\in\mathbb{R}^2$. Unfortunately no analytical expressions can be derived for these eigenvalues. So, we will use the perturbation analysis presented in Cortes[9] to prove the hyperbolicity of the eigenvalues and eigenvectors. We can see that the matrix $\mathbf{C}$ can be decomposed as follows

$$\mathbf{C} = \mathbf{C}_0 + \epsilon\ \mathbf{C}_1\ , \tag{18}$$

Where, the unperturbed or first approximation operator, C_0, has real eigenvalues and eigenvectors. Indeed, it can be easily proven that the following eigenvalue decomposition for $\mathbf{C}_0$ holds,

$$\mathbf{C}_0 = \mathbf{R}_0\ \Lambda_0\ \mathbf{R}_0^{-1}\ , \tag{19}$$

where the diagonal matrix Λ_0 has the eigenvalues of $\mathbf{C}_0$:

$$\begin{aligned} &\lambda_1 = \vec{V}_l\cdot\vec{n}, \quad \lambda_2 = \vec{V}_l\cdot\vec{n} + \sqrt{\alpha C_P\|\vec{V}_r\|^2}, \\ &\lambda_3 = \vec{V}_l\cdot\vec{n} - \sqrt{\alpha C_P\|\vec{V}_r\|^2}, \quad \lambda_4 = \vec{V}_g\cdot\vec{n}, \\ &\lambda_5 = \vec{V}_g\cdot\vec{n} + \sqrt{\tfrac{\gamma p}{\rho_g}}, \quad \lambda_6 = \vec{V}_g\cdot\vec{n} - \sqrt{\tfrac{\gamma p}{\rho_g}}, \end{aligned}$$

and the matrix $\mathbf{R}_0$ is formed by the right eigenvectors,

$$\mathbf{R}_0^1 = \begin{bmatrix} 0 \\ -n_y \\ n_x \\ 0 \\ 0 \\ 0 \end{bmatrix}, \quad \mathbf{R}_0^4 = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ -n_y \\ n_x \end{bmatrix},$$

$$\mathbf{R}_0^2 = \begin{bmatrix} (\lambda_4-\lambda_2)^2 - \frac{\gamma p}{\rho_g} \\ (u_l + n_x\sqrt{\alpha C_P\|\vec{V}_r\|^2})(\mathbf{R}_0^2)_1 \\ (v_l + n_y\sqrt{\alpha C_P}\|\vec{V}_r\|^2)(\mathbf{R}_0^2)_1 \\ \gamma p \\ \gamma p\ [u_g - n_x(\lambda_4-\lambda_2)] \\ \gamma p\ [v_g - n_y(\lambda_4-\lambda_2)] \end{bmatrix},$$

$$\mathbf{R}_0^3 = \begin{bmatrix} (\lambda_4-\lambda_3)^2 - \frac{\gamma p}{\rho_g} \\ (u_l - n_x\sqrt{\alpha C_P\|\vec{V}_r\|^2})(\mathbf{R}_0^3)_1 \\ (v_l - n_y\sqrt{\alpha C_P}\|\vec{V}_r\|^2)(\mathbf{R}_0^3)_1 \\ \gamma p \\ \gamma p\ [u_g - n_x(\lambda_4-\lambda_3)] \\ \gamma p\ [v_g - n_y(\lambda_4-\lambda_3)] \end{bmatrix},$$

$$\mathbf{R}_0^5 = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 1 \\ u_g + n_x\sqrt{\frac{\gamma p}{\rho_g}} \\ v_g + n_y\sqrt{\frac{\gamma p}{\rho_g}} \end{bmatrix}, \quad \mathbf{R}_0^6 = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 1 \\ u_g - n_x\sqrt{\frac{\gamma p}{\rho_g}} \\ v_g - n_y\sqrt{\frac{\gamma p}{\rho_g}} \end{bmatrix}.$$

proximation operator, we can apply the perturbation theory (Kato[19]) to conclude that: if the first approximation operator has real eigenvalues and eigenvectors, then for small ϵ, $\mathbf{C}$ has distinct real eigenvalues and the system of equations is hyperbolic. In this work, although we carry out the calculation of the fluxes with the exact matrix ($\mathbf{C}$), the distribution of the residual is made according the first approximation operator ($\mathbf{C_0}$).

Residual distribution schemes

Previously introduced by Roe, the Fluctuation Splitting or Residual Distribution schemes have reached nowadays a certain degree of maturity in applications involving the solution of steady non-linear systems of equations on unstructured grids. Several applications of these schemes to relevant engineering flows ranging from the Euler equations to the incompressible and compressible laminar and turbulent Navier-Stokes equations can be found in the literature.

Initially developed for the solution of scalar advection problems they have been succesfully extended to systems through the introduction of the matrix schemes (see E. van der Weide[20]). Here we will recall briefly the scalar formulation of the schemes and then describe their extension to linear and non-linear systems.

Short description of RDS

The residual distribution schemes (RDS) have been designed for an optimal discretization of the steady state conservation law

$$\vec{\lambda} \cdot \nabla u = 0, \tag{20}$$

on P1 finite element meshes, i.e. triangular (resp. tetrahedral) meshes with a piecewise linear solution representation u^h. We first define the cell residual or fluctuation over an element T as

$$\phi^T = \int_{\Omega_T} \vec{\lambda} \cdot \nabla u \; d\Omega. \tag{21}$$

Since ∇u^h is constant because of the linearity of the P1 element, an exact equivalence between the conservative form and the quasilinear form of the residual is found when the cell-averaged advection speed $\vec{\bar{\lambda}}$ is computed by the exact integration of

$$\vec{\bar{\lambda}} = \frac{1}{\Omega_T} \int_{\Omega_T} \vec{\lambda} \; d\Omega. \tag{22}$$

cell residual

$$\phi^T = \sum_i k_i u_i, \tag{23}$$

where $k_i = \frac{1}{2}\vec{\bar{\lambda}} \cdot \vec{n}_i$, termed the inflow parameter, and $\vec{n}_i$ is the scaled inward normal of the edge opposite to node i (see Fig. 1). The residual distribution method consists in distributing fractions of ϕ^T to the vertices of the element. Therefore, the resulting discrete equations express that the nodal residual ϕ_i^T, which consists of the sum of all contributions from neighbouring elements, vanishes,

$$\phi_i = \sum_{T \in \Delta_i} \phi_i^T = \sum_{T \in \Delta_i} \beta_i^T \phi^T. \tag{24}$$

In the above expression, β_i^T are the distribution coefficients and ϕ_i^T denotes the part of the residual ϕ^T of the element T which is sent to node i. On each element T, the distribution coefficients must add to one for consistency and conservativity (see Fig. 1).

Different schemes, corresponding to different ways of computing the distribution coefficients (β_i^T), have been designed to satisfy several properties like upwind character, positivity, linearity preservation and continuity. In this work we used the N-scheme and the LDA-scheme, defined as

- N-scheme.

$$\begin{array}{c} \beta_i^T = k_i^+ (\phi_T \sum_j k_j^-)^{-1} \sum_j k_j^- (u_i - u_j), \\ k_i^- = min(0, k_i), \quad k_i^+ = max(0, k_i). \end{array} \tag{25}$$

- LDA-scheme.

$$\beta_i^T = (\sum_j k_j^+)^{-1} k_i^+. \tag{26}$$

The N-scheme is a linear monotone first order upwind scheme, and the LDA-scheme is a second order, linear upwind scheme.

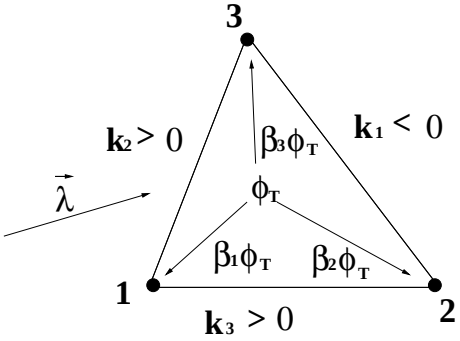

Fig. 1 Definition of inflow parameters

The residual distribution schemes has been also extended to unsteady scalar and vectorial equations. Consider the following 2-D hyperbolic system of partial differential equations (14):

$$\frac{\partial \vec{U}}{\partial t} + \mathbf{A}\frac{\partial \vec{U}}{\partial x} + \mathbf{B}\frac{\partial \vec{U}}{\partial y} = 0 \, .$$

The unknown vector $\vec{U}$ is assumed to vary linearly over each triangle like in the scalar case. The matrices $\mathbf{A}$ and $\mathbf{B}$ generally do not commute, i.e. they do not share the same right and left eigenvector matrices. The generalization to vectorial schemes is done in the following way:

The residual of a triangle T is defined as:

$$\vec{\Phi}^T = -\int_{\Omega_T} \frac{\partial \vec{U}}{\partial t}\, d\Omega = \int_{\Omega_T} (\mathbf{A}\frac{\partial \vec{U}}{\partial x} + \mathbf{B}\frac{\partial \vec{U}}{\partial y})\, d\Omega =$$

$$\left(\int_{\Omega_T} \mathbf{A}(\vec{U})\, d\Omega_T\right) \overline{\frac{\partial \vec{U}}{\partial x}} + \left(\int_{\Omega_T} \mathbf{B}(\vec{U})\, d\Omega_T\right) \overline{\frac{\partial \vec{U}}{\partial y}} =$$

$$\bar{\mathbf{A}}\,\overline{\frac{\partial \vec{U}}{\partial x}} + \bar{\mathbf{B}}\,\overline{\frac{\partial \vec{U}}{\partial y}} \tag{27}$$

where $\overline{\frac{\partial \vec{U}}{\partial x}}$ and $\overline{\frac{\partial \vec{U}}{\partial y}}$ are constant vectors since $\vec{U}$ varies linearly over T. The matrices $\bar{\mathbf{A}}$ and $\bar{\mathbf{B}}$ are the non-linear $\mathbf{A}$, $\mathbf{B}$ averaged over the domain Ω_T. The residual distribution discretization of system (14) is obtained by re-distributing the fluctuation of each triangle to its nodes using matrix distribution coefficients $|\mathbf{B}_i^T$, so the nodal residual, as well as for the scalar case, is obtained from the following relation:

$$\vec{\Phi}_i^T = \sum_{T \in \Delta_i} |\mathbf{B}_i^T \vec{\Phi}^T. \tag{28}$$

Consistency requires

$$\sum_{i \in T} |\mathbf{B}_i^T = \mathbf{I}. \tag{29}$$

where $\mathbf{I}$ is the identity matrix.

Since $\vec{U}$ varies linearly over each triangle the fluctuation $\vec{\Phi}^T$ can be shown to be given by

$$\vec{\Phi}^T = \sum_{i \in T} \mathbf{K}_i^T \vec{U}_i \, , \tag{30}$$

where the matrices $\mathbf{K}_i^T$ are the multidimensional generalization of the scalar inflow parameters k_i^T, and are given by

$$\mathbf{K}_i^T = \frac{1}{2}(\mathbf{A}n_x + \mathbf{B}n_y). \tag{31}$$

Two different problems appear in the application of these schemes to the isentropic two-fluid model:

No conservative linearization is available for system (14) and here we do not investigate the possibility of extending Roe-type linearizations to the two fluid model. This problem will be just reduced to the numerical quadrature of the integrals involved in the calculus of the residual. However, this is not a severe drawback, so according to Abgrall and Barth,[21] the selection of a numerical quadrature with the at least the same order of discretization error of the scheme is enough to assure conservation. In this work, mid-point rule has been used to compute the integrals involved in the residual.

- Upwinding using approximate eigenvalue decomposition:
  The schemes considered in this work are the N-scheme and the LDA-scheme. Since they both have the multidimensional upwinding property, some knowledge of the eigenstructure of the system is required. Although the model has been proven to be hyperbolic, no analytical expression can be derived for the eigenvalues of matrix (18), so the approximate eigenstructure presented in the previous section must be used to replace them. In order to overcome this point the following definitions has been taken into account:

  We have showed that the decomposition $\mathbf{C} = \mathbf{C}_0 + \epsilon\, \mathbf{C}_1$ holds. As the multi-dimensional inflow matrices are simply given by $\frac{1}{2}\mathbf{C}$, they can also be decomposed as

  $$\mathbf{K}_i = \mathbf{K}_{i_0} + \epsilon\, \mathbf{K}_{i_1}.$$

  Then the following $\tilde{\mathbf{K}}^{\pm}$ matrices are defined:

  $$\tilde{\mathbf{K}}_i^{\pm} = \frac{1}{2}\left(\mathbf{K}_i \pm |\mathbf{K}_{i_0}|\right) = \mathbf{K}_{i_0}^{\pm} + \epsilon \frac{\mathbf{K}_{i_0}}{2}. \tag{32}$$

  After this definition, the extension of the N-LDA schemes from the scalar schemes is straightforward.

  The N-scheme is defined as

  $$\vec{\Phi}_i = \tilde{\mathbf{K}}_i^{+}(\vec{U}_i - \vec{\tilde{U}}_{in}), \tag{33}$$

  where $\vec{\tilde{U}}_{in} = \left(\sum_{j \in T} \tilde{\mathbf{K}}_j^{-}\right)^{-1} \sum_{l \in T} \tilde{\mathbf{K}}_l^{-} \vec{U}_l$, which can be easily proven to be still conservative.

  On the other hand, the LDA-scheme is defined as:

  $$|\mathbf{B}_i^{LDA} = \mathbf{K}_i^{+} \left(\sum_{j \in T} \mathbf{K}_j^{+}\right)^{-1} . \tag{34}$$

Faucet problem

This numerical benchmark was proposed by Ransom. This test consists of a vertical water jet, contained within a cylindrical channel, that is accelerated under the action of gravity. At the initial stage, the pipe is filled with an uniform column of water surrounded by stagnant vapor, such that the void fraction is 0.2 and the column has an uniform velocity of 10 m/s and an uniform pressure of 10^5 Pa.
The boundary condition are specified velocities of 10 m/s for the liquid and 0 m/s for the vapor at the inlet and constant pressure at the outlet. The water faucet problem can be solved analytically. This analytical solution shows the development of a void wave propagating at liquid velocity. Once the void wave exits the pipe, a steady void profile is established. The calculation has been carried out until this steady state is reached.

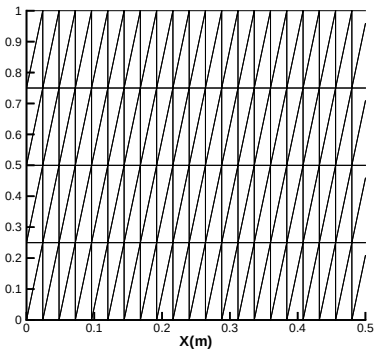

Fig. 2 Zoom on the 500×4 cells grid, used for the pseudo 1-D computations

In fact, this test is a real 1D problem, but in our work is solved with a 2D solver in a non-structured grid usig periodic boundary conditions (see figure 2).

Note that, this analytical solution is obtained from the basic (Wallis) model, so our model is expected to give a solution close to the analytical one if the value of C_P is very small. The influence of the value of C_P coefficient was investigated using the 1-D code. Different runs were made on a quite fine grid of 1000 points with C_P varying from 0.002 to 0.5. The obtained results are depicted in figure 3.

It can be seen that the higher is the value of C_P the more diffusive the model is. On the other hand, very small values of the interface pressure coefficient result in an unstable behaviour of the solution. In order to get a compromise the value of $C_p = 0.02$ was vergence to the exact solution can be seen in figure 4, while the time evolution of the void fraction, computed on a quite coarse grid (500 points), is in figure 5.

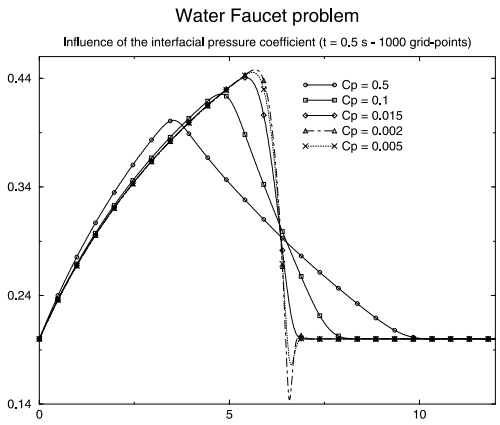

Fig. 3 Influence of the value of C_P on the solution

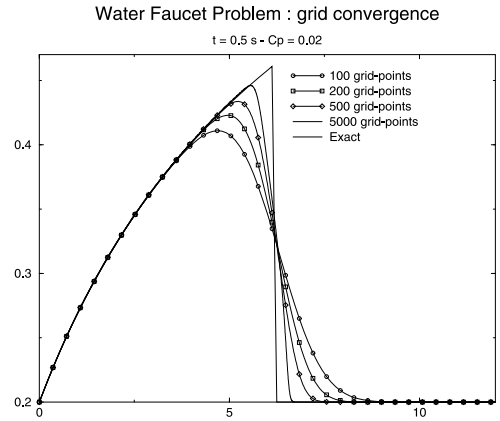

Fig. 4 Grid convergence for $C_P = 0.02$, $t = 0.5\ s$

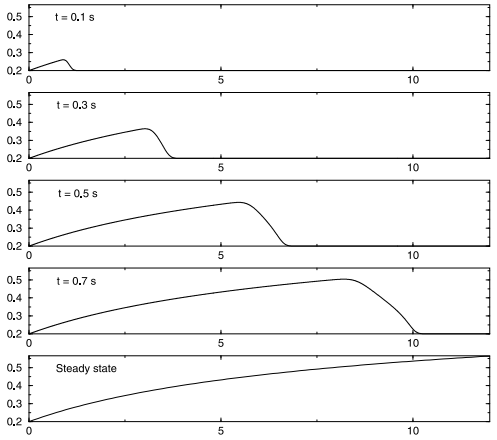

Fig. 5 Time evolution of the void fraction for $C_P = 0.02$, 500 grid-points

NACA0012 airfoil

Several major aviation accidents have occurred during extremely heavy rainfall. This have sparked an interest in heavy rain effect on airfoil and aircraft perfomance in rain. Experimental data indicate that an airfoil in heavy rain is subject to an overall degradation of performance (Bezos and Campbell,[22] Bezos et al.[23]). Numerical works in this area can be found in Valentine and Decker[24] where they applied a lagrangian-eulerian scheme for flow around an airfoil in rain. The influence of the water on the lift coefficient is shown for differents angles of attack. Of course, the loss of perfomance is bigger as soon as the angle of attack is increased, showing that the influcence on the boundary layer must be taked into account. In this work, the study is totally inviscid and even the drag between phases is neglected. For these reasons we only analise low angles of attack, where the effect of the boundary layer should be smaller.

On this conditions two different configuration have been tested:

- Configuration 1:

  $M_{g_\infty} = 0.5$; $\alpha_{g_\infty} = 0.8$; $U_{l_\infty} = \frac{1}{4} U_{g_\infty}$; $\beta_\infty = 5°$ (angle of attack) ;

- Configuration 2:

  $M_{g_\infty} = 0.5$; $\alpha_{g_\infty} = 0.9$; $U_{l_\infty} = \frac{1}{4} U_{g_\infty}$; $\beta_\infty = 5°$ (angle of attack).

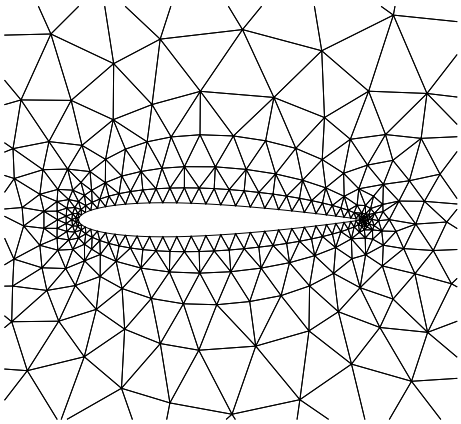

Fig. 6 Zoom view of the grid around the airfoil

The grid used in the computations is composed by 903 elements and 489 nodes, (figure 6). The LDA scheme is used in the all computations.

used:

$$p^* = \frac{p - p_\infty}{p_\infty}.$$

For the configuration 1, contours of p^* with gas streamlines superimposed are shown in figures 7 and 9, while contours of α_l with liquid streamlines superimposed in figure 8 and 10. It can be observed that the liquid is concentrated in the region around the leading edge. This is due to the higher inertia of the liquid phase which is driven by the pressure less than the gas phase, resulting in higher values of α_g close to the stagnation point. Finally, the computed pressure field for both configurations is shown in figures 11 and 12, together with the single phase Euler solution obtained with the VKI residual distribution solver IcARus. It is observed that, even though the differences are slight, as soon as the angle of attach is increased the loss of perfomance of the airfoil becomes more significant.

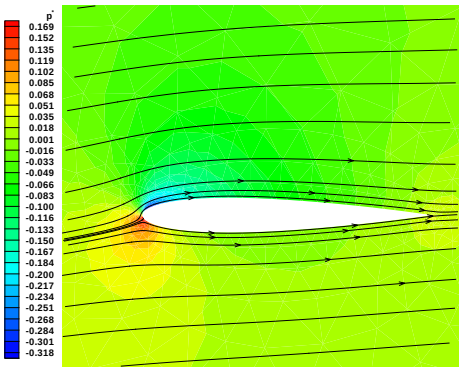

Fig. 7 p^* contours and gas streamlines for the configuration 1

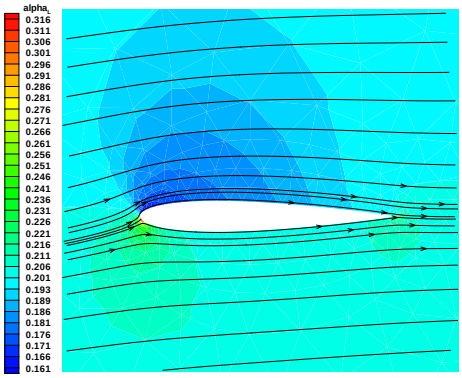

Fig. 8 α_l contours and liquid streamlines for the configuration 1

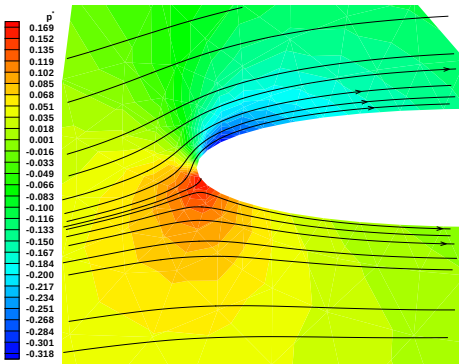

Fig. 9 Close-up view of the p^* contours and gas streamlines at the border of attack in the configuration 1

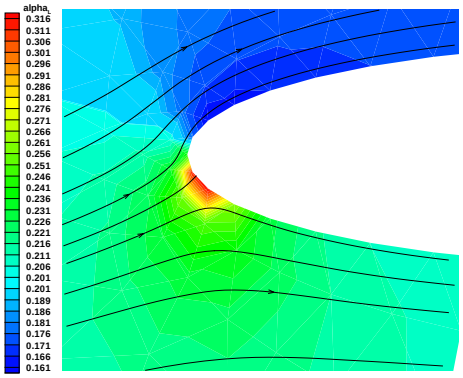

Fig. 10 Close-up view of the α_l contours and liquid streamlines at the border of attack in the configuration 1

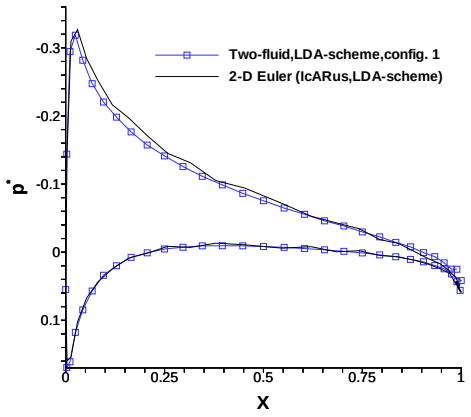

Fig. 11 Comparison of the p^* profile with a 2-D Euler computation for configuration 1

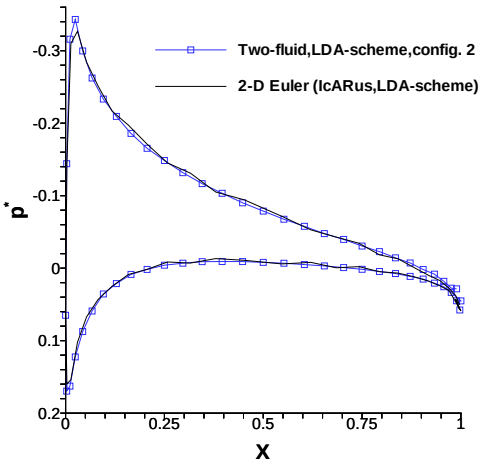

Fig. 12 Comparison of the p^* profile with a 2-D Euler computation for configuration 2

Conclusion

Starting form a simple isentropic two-fluid model a residual distribution 2-D code for two-phase flow computations has been developed. The hyperbolicity of the system of equations was shown using the perturbation theory and a formal extension of both the N-scheme and the LDA-scheme to systems, for which the exact eigenstructure is unknown, was given. Although the numerical results were satisfactory, several issues still remain open.

The linearization of the equations is obtained just applying a mid-point quadrature rule to the integrals that appear in the calculus of the residual. A better approach should be necessary for the simulation of flows involving strong discontinuities.

The current approach cannot manage the transition from one phase to another and slip ratios tending to zero. To be exact, although the system is valid in the limits $\alpha_g \to 1$, $\alpha_g \to 0$, $\vec{V}_r \to 0$, from a mathematical point of view it is ill-posed.

In this work, upwinding along directions given by an approximate eigenstructure of the system has shown satisfactory results. But the possibility of working with fully hyperbolic models (like that proposed by Stadtke[25]), for which the analytical form of the eigenvalues and eigenvectors is known, still remains very attractive.

References

[1] M. Ishii. *Thermo-Fluid Dynamic Theory of Two-Phase Flow*. Collection de la Direction des Études

1975.

[2] D. A. Drew. Mathematical modeling of two-phase flow. *Ann. Rev Fluid Mech.*, 15:261–291, 1983.

[3] G. F. Hewitt, J. M. Delhaye, and N. Zuber, editors. *Two-phase flow fundamentals*, volume 8 of *Multiphase Science and Technology*, chapter 1. Begell House, 1994.

[4] S. Banerjee and A. M. C. Chan. Separated flow models – I. *Int. J. Multiphase Flow*, 6:1–24, 1980.

[5] H. B. Stewart. Two-Phase Flow: Models and Methods. *J. Comp. Phys.*, 56:363–409, 1984.

[6] V. H. Ranson and D. L. Hicks. Hyperbolic two-pressure models for two-phase flow. *J. Comp. Phys.*, 53:124–151, 1984.

[7] R. Courant and D. Hilbert. *Methods of Mathematical Physics*, volume II. Wiley-Interscience, Inc., New-York, 1962.

[8] I. Toumi and A. Kumbaro. An Approximate Linearized Riemann Solver for a Two-Fluid Model. *J. Comp. Phys.*, 124:286–300, 1996.

[9] J. Cortes, A. Debussche, and I. Toumi. A density perturbation method to study the eigenstructure of two-phase flow equation systems. *J. Comp. Phys.*, 147:463–484, 1998.

[10] M. Arai. Characteristics and stability analyses for two-phase flow equation systems with viscous terms. *Nuc. Scienc. Eng*, 74:77–83, 1980.

[11] H. Pokharna, M. Mori, and V. H. Ransom. Regularization of two-phase flow models: A comparison of numerical and differential approaches. *J. Comp. Phys.*, 134:282–295, 1997.

[12] H. Paillere. Multidimensional upwind residual distribution schemes for the Euler and Navier–stokes equations on unstructured grids. *PhD thesis, Universite Libre de Bruxelles, Belgium*, 1995.

[13] I. Toumi. An upwind numerical method for two-fluid two-phase flow models. *Nuc. Scienc. Eng*, 123:147–168, 1996.

[14] G. F. Hewitt, J. M. Delhaye, and N. Zuber, editors. *Numerical Benchmark tests*, volume 3 of *Multiphase Science and Technology*, chapter 3. Springer–Verlang, 1987.

the numerical solution to two-fluid models via a cell centered finite volume method. *to be published*, 2000.

[16] I. Tiselj and S. Petelin. Modelling of two-phase flow with second-order accurate scheme. *J. Comp. Phys.*, 136:503–521, 1997.

[17] A. Kumbaro I. Toumi and H. Paillere. Approximate riemann solvers and flux vector splitting schemes for two-phase flow. 30^{th} *VKI Lecture Series in Computational Fluid Dynamics*, 1999.

[18] R. T. Lahey. *Boiling Heat Transfer; Modern Developments and Advances.* Elsevier Science Publishers, New York, 1992.

[19] T. Kato. *Perturbation theory for linear operators.* Springer–Verlang, 1984.

[20] E. van der Weide. Compressible flow simulation on unstructured grids using multi-dimensional upwind schemes. *PHD thesis, Technical University of Delft*, 1998.

[21] Remi Abgrall and Timothy Barth. Residual distribution schemes for conservation laws via adaptive quadrature. *Support de Cours: Schémas distribués en mécanique des fluides et applications (Inria-Rocquencourt)*, pages 183–212, 2000.

[22] G.M. Bezos and B.A. Campbell. Development of a large-scale, outdoor, ground-based test capability for evaluating the effect of rain on airfoil lift. *NASA TM-4420*, 1993.

[23] G.M. Bezos, R.E. Dunham, G.L. Gentry, and W. E. Jr Melson. Wind tunnel aerodynamic characteristics of a transport-type airfoil in a simulated heavy rain environment. *NASA TP-3184*, 1992.

[24] J.R. Valentine and R.A. Decker. A lagrangian-eulerian scheme for flow around an airfoil in rain. *Int. J. Multiphase Flow*, 21(4):639–648, 1995.

[25] H. Stadtke, G. Franchello, and B. Worth. Numerical simulation of gas-liquid flows using flux vector splitting techniques. 8^{th} *Int.Symposium on Computational Fluid Dynamics, Bremen (Germany)*, 1999.