NETWORKS AND HETEROGENEOUS MEDIA
©American Institute of Mathematical Sciences
Volume **5**, Number **3**, September **2010**
doi:10.3934/nhm.2010.5.649
pp. **649–660**

SOME ATTEMPTS TO COUPLE DISTINCT FLUID MODELS

Jean-Marc Hérard and Olivier Hurisse
EDF R&D, MFEE, 6, quai Watier
Chatou, 78400, France

Abstract. We present in this paper a review of some recent works dedicated to the numerical interfacial coupling of fluid models. One main motivation of the whole approach is to provide some meaningful methods and tools in order to compute unsteady patterns, while using distinct existing CFD codes in the nuclear industry. Thus, the main objective is to derive suitable boundary conditions for the codes to be coupled. A first section is devoted to a review of some attempts to couple: (i) 1D and 3D codes, (ii) distinct homogeneous two-phase flow models, (iii) fluid and porous models. More details on numerical procedures described in this section can be found in companion papers. Then we detail in a second section a way to couple a two-fluid hyperbolic model and an homogeneous relaxation model.

1. **Introduction.** We present in this paper a review of recent investigations carried on within the framework of the NEPTUNE project ([21]). These aim at improving the interfacial coupling of distinct existing CFD codes, for industrial purposes, focusing on *unsteady* situations. Since most of the codes involved in this project provide approximations of two-phase flow models, special emphasis is given on this class of models.

Throughout the paper, we consider some computational domain $\Omega = \Omega_L \cup \Omega_R$ in $\mathcal{R}^2$, where $\Omega_L = ((x,y)/x < 0)$ and $\Omega_R = ((x,y)/x > 0)$, and a *coupling interface* standing at $x = 0$. We will focus on models of the form:

$$\frac{\partial W_L}{\partial t} + \frac{\partial (f_L(W_L))}{\partial x} + \frac{\partial (g_L(W_L))}{\partial y} = 0 \quad \text{for:} \quad x < 0,$$

and:

$$\frac{\partial W_R}{\partial t} + \frac{\partial (f_R(W_R))}{\partial x} + \frac{\partial (g_R(W_R))}{\partial y} = 0 \quad \text{for:} \quad x > 0,$$

where the state variables W_L and W_R are distinct and lie in $\mathcal{R}^p$ and $\mathcal{R}^q$ respectively. The two fluxes $f_L(W_L)$ and $g_L(W_L)$ (resp. $f_R(W_R)$ and $g_R(W_R)$) also lie in $\mathcal{R}^p$ (resp. in $\mathcal{R}^q$). In some cases, the flux g_L or g_R will be set to zero, when restricting to the one-dimensional framework. We insist that no thickening of the coupling interface is used here. Hence, the main problem here is to prescribe suitable boundary conditions on both sides of the coupling interface $x = 0$, in order to perform unsteady computations. Roughly speaking, we will enforce the continuity of some specific quantities to be defined through the coupling interface. This will be achieved thanks to a hyperbolic father model. The latter will naturally arise in some cases (see for instance sections 2.2, 2.3 and 3); however, in a few cases,

2000 *Mathematics Subject Classification.* Primary: 76T10, 65M08; Secondary: 76S05, 35Q70.
Key words and phrases. Two-phase flow models, Interfacial coupling, Unsteady flows, Riemann solver, Finite Volumes, CFD codes.

some alternative propositions may occur (see section 2.1 for instance). Once the father model has been chosen, boundary conditions for $x/t = 0^-$ and $x/t = 0^+$ are obtained in a straightforward manner, using the structure of the solution of the one-dimensional Riemann problem associated with the father model. The problem is indeed difficult, even in the scalar case (see [6, 9, 18, 30] among others). One may for instance exhibit quite easily situations where the coupled Riemann problem is ill-posed. However, we have to build stable coupling procedures for industrial applications and thus we need to study the case of systems. Most examples examined herein rely on models arising from the two-phase literature, and these will be detailed in each section.

The first section is devoted to a review of some experiments on: (i) the numerical coupling of 1D and 2D models, (ii) the simulation of flows coming from a free medium and entering a porous medium, (iii) the coupling of two simple distinct two-phase flow models. The second section examines the problem of the numerical coupling of two wide-spread two-phase flow models, namely the homogeneous relaxation model and the two-fluid model. The list of references is obviously not exhaustive, and we will insist on major conclusions, unexpected results, drawbacks and weaknesses of the current work on this topic.

2. **A review of some interfacial coupling experiments.**

2.1. **Coupling 1D and 2D models.**

The work described in [26] is basically motivated by the fact that many CFD codes have been developped in the past, making the assumption that flows in pipes may be represented by a pure one-dimensional framework.

Thus, the left code provides approximations of the solutions of the governing set of equations:

$$\frac{\partial W_L}{\partial t} + \frac{\partial (f_L(W_L))}{\partial x} = 0, \tag{1}$$

in $\Omega'_L = ((x,0)/x < 0)$. We may set here for instance: $f_L(W_L) = (\rho u, \rho u^2 + P, u(E_L + P))$, where $W_L = (\rho, \rho u, E_L)$ and ρ, u, P, E_L respectively stand for the state variable, the density, the velocity, the pressure and the total energy of the fluid. The total energy is $E_L = \rho(e(P,\rho) + u^2/2)$, and we get a closed set of equations once the equation of state (EOS) $e(P,\rho)$ has been provided by users.

Meanwhile, the flow properties in the right code, for $(x,y) \in \Omega_R$, issue from:

$$\frac{\partial W_R}{\partial t} + \frac{\partial (f_R(W_R))}{\partial x} + \frac{\partial (g_R(W_R))}{\partial y} = 0. \tag{2}$$

The right state variable is: $W_R = (\rho, \rho u, \rho v, E_R)$, right fluxes are $f_R(W_R) = (\rho u, \rho u^2 + P, \rho uv, u(E_R + P))$ and $g_R(W_R) = (\rho v, \rho uv, \rho v^2 + P, v(E_R + P))$, where v denotes the second velocity component, which has been set to 0 in the left code (and thus omitted). Besides, E_R is defined as $E_R = \rho(e(P,\rho) + (u^2+v^2)/2) = E_L + \rho v^2/2$.

The coupling technique requires to define in a suitable way left and right boundary numerical fluxes for the interface connecting left and right codes. Two different attempts have been made up to now: the first one is conservative with respect to the momentum equation, the second one is not. These experiments have been achieved assuming in each case that a prescribed father model governs evolutions of the fluid around the coupling interface. Defining $Z(x,y,t) = 0$ for $(x,y) \in \Omega_L$

and $Z(x,y,t) = 1$ for $(x,y) \in \Omega_R$, the conservative father model CFM is:

$$\frac{\partial W}{\partial t} + \frac{\partial (h(W))}{\partial x} = 0, \tag{3}$$

where $W = (Z, \rho, \rho u, \rho v)$, and $h(W) = (0, \rho u, \rho u^2 + P, Z\rho uv)$. The alternative non conservative father model $NCFM$ reads:

$$\begin{aligned} \frac{\partial Z}{\partial t} &= 0, \\ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} &= 0, \\ \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2 + P)}{\partial x} &= 0, \\ \frac{\partial v}{\partial t} + Zu\frac{\partial v}{\partial x} &= 0. \end{aligned} \tag{4}$$

Once the father model has been chosen among the two candidates (3), (4), left and right coupling fluxes are fully determined by solving the one-dimensional Riemann problem associated with (3), or (4) either. Left and right fluxes are of course equal in the conservative case.

An extensive investigation of these two approaches has been achieved, and a comparison with the results issuing from (2) on the whole computational domain $\Omega = \Omega_L \cup \Omega_R$, is available in [26] and associated references. Numerical experiments have been achieved while connecting a one-dimensional pipe with a constant cross-section to a two-dimensional tank across the interface $x = 0$. Rarefaction waves and shocks were generated within each subdomain, and their propagation through the coupling interface was observed and compared with a reference two-dimensional simulation on the whole computational domain $\Omega_L \cup \Omega_R$, while using a very fine mesh. We can briefly summerize the main conclusions as follows:

- In some cases, the conservative father model may lead to a blow up of the code, especially when trying to refine the mesh. This is actually an unexpected -and unexplained- weakness of the conservative approach. Moreover, a better agreement with the two-dimensional reference solution over the whole domain has been noticed when using the non-conservative model (4).
- The position of the coupling interface plays a key role, as expected by engineers. Obviously, the coupling interface should be put far enough inside the left domain Ω_L that supports the one-dimensional approach. Otherwise, results of the coupled computations may be quite different from those provided by the pure $2D$ approach, which is indeed the reference solution.
- The quality of results hardly depends on the liquid or gas EOS, which could not be guessed *a priori*.
- The approximate Riemann solver which is used in practice at the coupling interface has little influence on the computational results.

2.2. **Handling the transition between free and porous media.** Many nuclear codes and applications for the nuclear power reactors rely on the use of the so-called porous approach (see [21]). This means for instance that all tubes in steam generators are not accounted for, since a control volume may contain many obstacles, even in a refined mesh. The porosity $\epsilon(x,y,z)$ simply stands for the ratio occupied by the fluid $\mathcal{V}_f$ over the total controle volume $\mathcal{V}$. This is refered to as the porous approach.

One crucial problem then may be the following: what happens when the mean incoming flow in a free region enters a porous region (where the porosity ϵ is usually almost uniform, and at least may be assumed smooth enough) ? Actually, the latter problem is rather tough, for at least two main reasons. At first, it is known from long that sharp free/porous transitions may lead to oscillations which take their roots in the formulation of *schemes* (see [29]). A second point is that the coupling conditions around the coupling interface are not clearly written in the literature. We only focus here on the first item, thus assuming that some suitable "coupling interface" boundary conditions have been prescribed.

Up to now, two different approaches have been investigated. The first approach relies on the two-phase homogeneous approach (see [23]). The second one, which requires a specific formulation (see [24]), is based on the two-fluid formulation (see [15]).

2.2.1. *The homogeneous approach.* We restrict here to the one-dimensional and isentropic case [23]. We define: $W_L = (1, \rho, \rho u)$, $f_L(W_L) = (0, \rho u, \rho u^2 + P(\rho))$, $g_L(W_L) = (0, 0, 0)$ and also $W_R = (\epsilon, \epsilon\rho, \epsilon\rho u)$, $f_R(W_R) = (0, \epsilon\rho u, \epsilon\rho u^2)$, $g_R(W_R) = (0, 0, 0)$ and $h_R(W_R) = (0, 0, P(\rho))$. We emphasize that the right model has no longer a conservative form. Thus, the two models to be coupled are now:

$$\frac{\partial W_L}{\partial t} + \frac{\partial (f_L(W_L))}{\partial x} = 0, \tag{5}$$

in Ω_L, and:

$$\frac{\partial W_R}{\partial t} + \frac{\partial (f_R(W_R))}{\partial x} + \epsilon \frac{\partial (h_R(W_R))}{\partial x} = 0, \tag{6}$$

in Ω_R. Since the porosity does not vary with time, the hyperbolic system (6) admits a standing wave (corresponding to the eigenvalue $\lambda = 0$), and we note $I_0^k(W_R)$ the associated Riemann invariants (for $k = 1, 2$). In order to perform the coupling of (5) and (6), it has been assumed that the boundary conditions at the coupling interface are directly deduced from the preservation of the Riemann invariants of the standing wave associated with the father model corresponding to system (6), that is:

$$I_0^k(W_L^-) = I_0^k(W_R^+).$$

We have noted here $W_L^- = W_L(x/t = 0^-)$, and $W_R^+ = W_R(x/t = 0^+)$. When focusing on the isentropic Euler equations, we get $I_0^1(W_R) = \rho\epsilon U$ and $I_0^2(W_R) = U^2/2 + \psi(\rho)$, while setting $\rho\psi'(\rho) = c^2(\rho)$. This actually corresponds to the basic ideas of Greenberg and Leroux (see [20]). Given these boundary conditions, we may now perform the numerical coupling of the two models. This may be achieved by defining a Finite Volume scheme that gives approximate solutions of (6), and then setting $\epsilon(x) = 1$ for x in Ω_L, and: $\epsilon(x) = \epsilon_0(x) < 1$ for x in Ω_R. Two schemes have been defined and investigated in [23] and [22]. These two schemes aim at enforcing at each interface the continuity of the above-mentionned Riemann invariants $I_0^k(W_R)$. In [23], we have used the approximate Godunov scheme VFRoe-ncv introduced in [8], using the non-conservative variable $Z = (I_0^1(W_R), I_0^2(W_R))$. The same idea has been used in [22] for the non-isentropic Euler equations in a porous medium. These two schemes are well-balanced in the sense of [20], which means that they perfectly maintain *all* steady states whatever the mesh size is. Actually, numerical results that have been obtained in the isentropic and non-isentropic case are fair.

2.2.2. *The two-fluid approach.* This problem is again motivated by the existing codes Cathare and Neptune-CFD for instance (see [21]), and we refer to [15, 16] for details. The equations solved in both codes use the two-fluid approach, and the former one may account for porous regions.

The coupling of the two formulations has been investigated with a slightly different insight, assuming that the instantaneous pressure relaxation step implicitely contained in these codes is skipped. Hence the unknown of the two-fluid model involves two distinct pressure fields (one for each phase). Otherwise, initial-value problems may become ill-posed, even for small relative velocities. The index k will refer to the phase number, and the subscript i to the interface quantities. Hence, the father model that has been examined is (see [24] that introduces the model and its main properties):

$$\left\{\begin{array}{l} \dfrac{\partial \epsilon}{\partial t} = 0, \\ \dfrac{\partial \alpha_k}{\partial t} + V_i \dfrac{\partial \alpha_k}{\partial x} = \phi_k, \\ \dfrac{\partial (\epsilon m_k)}{\partial t} + \dfrac{\partial (\epsilon m_k U_k)}{\partial x} = 0, \\ \dfrac{\partial (\epsilon m_k U_k)}{\partial t} + \dfrac{\partial (\epsilon m_k U_k^2)}{\partial x} + \epsilon \alpha_k \dfrac{\partial P_k}{\partial x} + \epsilon (P_k - P_i) \dfrac{\partial \alpha_k}{\partial x} = \epsilon D_k, \\ \dfrac{\partial (\epsilon \alpha_k E_k)}{\partial t} + \dfrac{\partial (\epsilon \alpha_k U_k (E_k + P_k))}{\partial x} + \epsilon P_i \dfrac{\partial \alpha_k}{\partial t} = 0, \end{array}\right. \tag{7}$$

where ρ_k, $\alpha_k \in [0,1]$, m_k, U_k, P_k, $e_k(P_k, \rho_k)$ and $E_k = \rho_k U_k^2/2 + \rho_k e_k(P_k, \rho_k)$ respectively denote the density, the volume fraction, the mass fraction, the velocity, the pressure, the internal energy and the total energy within phase k, assuming that $\alpha_1 + \alpha_2 = 1$. The couple (P_i, V_i) has been set to (U_2, P_1), according to [7, 12, 14]. The closure laws for the source terms are the following:

$$\left\{\begin{array}{l} D_k = m_1 m_2 K_u (U_{3-k} - U_k)/(m_1 + m_2), \\ \phi_k = \alpha_1 \alpha_2 K_P (P_k - P_{3-k})/\Pi_0, \end{array}\right. \tag{8}$$

for $k = 1, 2$, where K_u, K_P are two positive functions depending on the "conservative" variable W, and correspond to the inverse of time scales, while Π_0 denotes some reference value of the mean pressure.

The problem now is similar to the one discussed in the previous section 2.2.1. We need to couple the flow in a free medium (corresponding to $\epsilon(x) = 1$ in Ω_L), and the flow in a porous medium (corresponding to a given distribution $\epsilon(x) < 1$ in Ω_R). As in the previous case, the boundary conditions at the coupling interface correspond to the preservation of $0-$Riemann invariants. We emphasize that exact solutions of the one-dimensional Riemann problem associated with system (7) may be exhibited, that allow a straightforward measure of the convergence rate.

The main conclusions that arise from the preliminary work [23, 15, 16, 22] are the following:

- Simple and expected suitable schemes, which are not well-balanced, may converge towards a wrong solution, when a flow encounters a sudden porosity transition. In figure 1, we show the L^1 norm of the error when using Rusanov scheme and computing such a Riemann problem with discontinuous porosity; in that case the coarser mesh and the finer mesh contain 100 and 400000 regular cells respectively, and the CFL number has been set to 1/2; initial conditions and exact values of intermediate states can be found in [15]. Actually,

as underlined in [15], the well-balanced criterium seems to be mandatory, in order to obtain a correct convergence. A specific formulation that is inspired by the well-balanced Rusanov scheme introduced in [29] provides a good candidate, though of course the $h^{1/2}$ rate of convergence remains rather weak for practical applications. Evenmore, the approach suggested in [22], that is also grounded on the well-balanced techniques, but which relies on an approximate Godunov flux rather than a Rusanov flux, constitutes a fair improvement of the latter well-balanced Rusanov scheme, in terms of accuracy.

- Mathematical coupling boundary conditions associated with the basic approach of the father model (7), as implicitely suggested by [20], may certainly be improved in the 2D or 3D framework (see [16]). More precisely, the enforcement of two particular $0-$Riemann invariants (namely s_1, s_2 where s_k denotes the specific entropy within phase k) of the standing wave in the well-balanced formulation might certainly be replaced by two more relevant physical conditions. This obviously urges some deeper investigation, and some basic ideas are now examined.
- Beyond this point, and though it seems to be rather crude, the so-called porous approach may be useful for industrial purposes. Actually, a rather interesting point is that it may converge much faster than the free approach -in terms of mesh refinement-, at least in some configurations (see [16]).

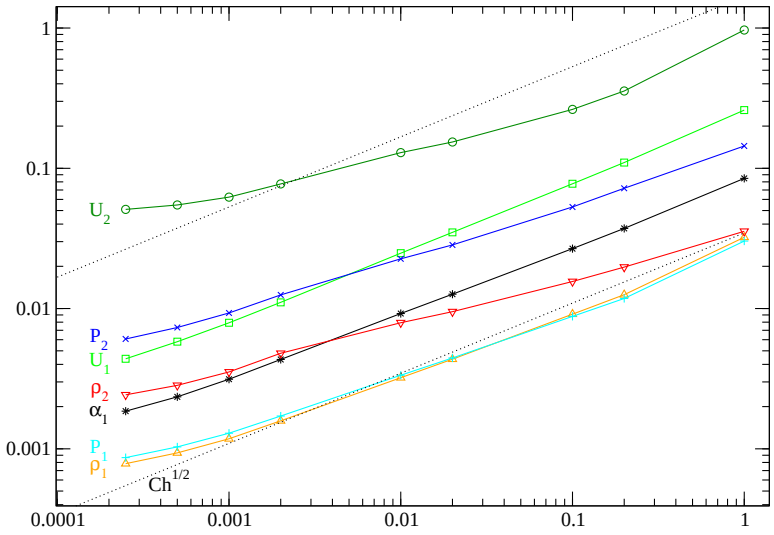

FIGURE 1. L^1 norm of the error when computing a one-dimensional Riemann problem with a discontinuous profile of the porosity ϵ, while using a standard scheme (Rusanov scheme).

2.3. **The numerical coupling of HEM and HRM models.** This subsection summarizes part of the work detailed in [5] (see [2] also where the drift velocity is

taken into account). HEM (Homogeneous equilibrium model) and HRM (Homogeneous relaxation model) are two widespread models that are used by the nuclear community. Both models assume some *local* closure for the relative velocity between phases $U_2 - U_1$, but also on the pressure and temperature differences between phases. The HRM model does not assume a Gibbs potential equilibrium between both phases, and a straightforward consequence is that a mass transfer term Γ is present in the governing equations of the mass balance of both phases (see (9)). The state variable of the HRM model is:

$$Z_{hrm} = (X, \rho_m, U_m, P_m),$$

where: $\rho_m = \alpha_1\rho_1 + \alpha_2\rho_2$, $X = \alpha_2\rho_2/\rho_m$, $U_m = (\alpha_1\rho_1U_1 + \alpha_2\rho_2U_2)/\rho_m$, $P_m = \alpha_1P_1 + \alpha_2P_2$, and $E_m = \alpha_1E_1 + \alpha_2E_2$. Assuming in addition that a velocity-temperature-pressure equilibrium holds everywhere in Ω_R, eg: $U_1 - U_2 = 0$, $T_1 - T_2 = 0$, $P_1 - P_2 = 0$, the governing set of equations of the HRM model reads:

$$\begin{cases} \dfrac{\partial(\rho_m X)}{\partial t} + \dfrac{\partial(\rho_m X U_m)}{\partial x} = \Gamma, \\ \dfrac{\partial \rho_m}{\partial t} + \dfrac{\partial(\rho_m U_m)}{\partial x} = 0, \\ \dfrac{\partial(\rho_m U_m)}{\partial t} + \dfrac{\partial(\rho_m U_m^2)}{\partial x} + \dfrac{\partial P_m}{\partial x} = 0, \\ \dfrac{\partial E_m}{\partial t} + \dfrac{\partial(U_m(E_m + P_m))}{\partial x} = 0, \end{cases} \tag{9}$$

where $E_m = \rho_m(e_m(P_m, \rho_m, X) + U_m^2/2)$. The state variable in the HEM model is noted $Z_{hem} = (\rho, U, P)$, and Z_{hem} complies with:

$$\begin{cases} \dfrac{\partial \rho}{\partial t} + \dfrac{\partial(\rho U)}{\partial x} = 0, \\ \dfrac{\partial(\rho U)}{\partial t} + \dfrac{\partial(\rho U^2)}{\partial x} + \dfrac{\partial P}{\partial x} = 0, \\ \dfrac{\partial E}{\partial t} + \dfrac{\partial(U(E + P))}{\partial x} = 0, \end{cases} \tag{10}$$

where $E = \rho(e(P, \rho) + U^2/2)$; ρ_m, P_m, U_m (respectively ρ, P, U) denote the mean density, the mean pressure and the mean velocity of the mixture.

The problem of the numerical coupling of HEM and HRM models in Ω_L and Ω_R respectively has been investigated in the pioneering work [2] and [5]. The previous references provide two slightly different insights on this coupling problem. The concept of a father model has once again been retained in the work [5]. Some conclusions may be drawn as follows.

- As pointed out in [2, 5], a first crucial point is that the two models should contain some inner coherence. More precisely, the consistency of the EOS of the HEM with the mass exchange term Γ in the HRM plays a key role in the behaviour of coupled simulations.
- Another probably more obvious remark pertains to the amplitude of the relaxation time involved in this source term Γ. When the latter is sufficiently small, the numerical coupling experiments exhibit little pollution around the steady coupling interface between the two codes.
- Eventually, it seems worth noting that the particular schemes that are involved at the coupling interface seem to have little influence on the quality of results.

3. **The numerical coupling of a two-fluid model with HRM in a free medium.** This section is devoted to some recent numerical attempts (see [25, 27]) to couple two different two-phase flow codes relying on the two-fluid approach (in Ω_L) and the homogeneous approach (in Ω_R) respectively. We provide here a brief description of the overall problem and techniques. The left code is assumed to provide approximations of the state variable:

$$Z_{tfm} = (\alpha_2, \rho_1, \rho_2, U_1, U_2, P_1, P_2),$$

using some given numerical method, such as the one described in [14]. The continuous model is exactly given by (7), setting $\epsilon(x) = 1$ everywhere in the computational domain, that is:

$$\begin{cases} \dfrac{\partial \alpha_k}{\partial t} + V_i \dfrac{\partial \alpha_k}{\partial x} = \phi_k, \\ \dfrac{\partial m_k}{\partial t} + \dfrac{\partial (m_k U_k)}{\partial x} = 0, \\ \dfrac{\partial (m_k U_k)}{\partial t} + \dfrac{\partial (m_k U_k^2)}{\partial x} + \alpha_k \dfrac{\partial P_k}{\partial x} + (P_k - P_i) \dfrac{\partial \alpha_k}{\partial x} = D_k, \\ \dfrac{\partial (\alpha_k E_k)}{\partial t} + \dfrac{\partial (\alpha_k U_k (E_k + P_k))}{\partial x} + P_i \dfrac{\partial \alpha_k}{\partial t} = 0. \end{cases} \tag{11}$$

Meanwhile, the right code generates approximations of

$$Z_{hrm} = (X, \rho_m, U_m, P_m),$$

in Ω_R, using notations introduced in the preceeding section 2.3, and where Z_{hrm} is a solution of the HRM model (9).

We point out that we have once more used the non-conservative father model (11) in order to perform the numerical coupling of the two models. This means that we solve the Riemann problem associated with the father model (11) together with the initial conditions provided by the left state $(Z_{tfm})_L$ and the right state $\mathcal{E}((Z_{hrm})_R)$. The latter state $(Z_{tfm})_R = \mathcal{E}((Z_{hrm})_R)$ in $\mathcal{R}^7$ is obtained by prescribing the local equilibria $(U_1)_R = (U_2)_R = (U_m)_R$, $((P_1)_R = (P_2)_R = (P_m)_R$, $(T_1)_R = (T_2)_R = (T_m)_R$ in the fictitious state $(Z_{tfm})_R$. Thus we set: $\mathcal{E}((Z_{hrm})_R) = \alpha_2, \rho_1(P_m, T_m), \rho_2(P_m, T_m), U_m, U_m, P_m, P_m)$ where α_2 complies with:

$$\alpha_2 \rho_2(P_m, T_m) = \rho_m X.$$

This enables us to define boundary conditions for both codes, as explained with more details in [27]. These boundary conditions guarantee the strict conservation of the total mass, the total momentum and total energy of both phases in the coupled simulation.

In order to illustrate this kind of coupling, we show in figures 2 and 3 the behaviour of all variables at time $t = 0.000877$. The initial conditions in the two-fluid domain have been chosen in such a way that: $T_1 = T_2$, $P_1 = P_2$ and $U_1 = U_2$. A perfect gas EOS holds within both liquid and vapour phases: $P_k = (\gamma_k - 1)\rho_k e_k$, with: $\gamma_1 = 1.1$, $\gamma_2 = 1.4$. Initial conditions on each side of x_r are: $P_1(x < x_r, t = 0) = 155 \times 10^5$, $U_1(x < x_r, t = 0) = 0$, $\rho_1(x < x_r, t = 0) = 800$, $\alpha_1(x < x_r, t = 0) = 0.995$ for $x < x_r$, and: $P_1(x > x_r, t = 0) = 150 \times 10^5$, $U_1(x > x_r, t = 0) = 0$, $\rho_1(x > x_r, t = 0) = 700$, $\alpha_1(x > x_r, t = 0) = 0.005$. The right-going shock wave that has been generated at $x = x_r = -0.25$ at the beginning of the computation propagates and hits the coupling interface $x = 0$, thus inducing reflected and transmitted waves.

We observe that the numerical pollution generated by the right-going waves coming from Ω_L and hitting the coupling interface tends to diminish when the mesh is refined. Unfortunately, we have no exact solutions here to assess the current methodology, unlike in previous sections 2.1 and 2.2. This renders the validation task more difficult, since we can only compare the coupled solution to the two solutions obtained when using either (11) or (9) over the whole domain Ω.

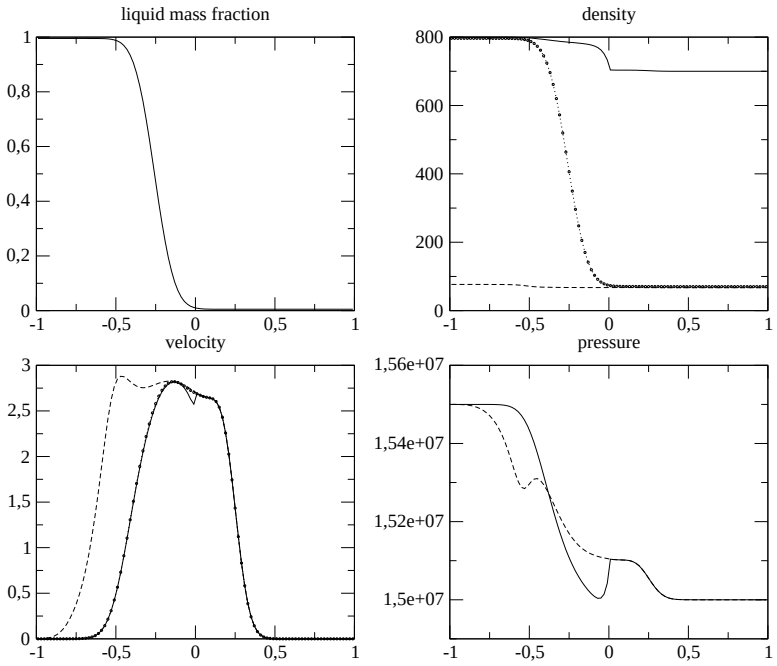

FIGURE 2. Interaction of a right-going shock wave, coming from the two-fluid region ($x < 0$), with the steady coupling interface at $x = 0$, the homogeneous region being on the right side ($x > 0$). The initial discontinuity of the Riemann problem was set at $x_r = -0.25$. The mesh contains 100 regular cells, and the CFL number is 1/2. Liquid phase: plain line / vapour phase: dashed line. A dotted line with circles has been used for mixture variables ρ_m, U_m.

A somewhat different technique has been proposed recently in [1], in order to achieve the same kind of interfacial coupling, though no longer assuming a zero relative velocity in the HRM. Since the model investigated in [1] does not take energy equations into account (each phase is assumed to comply with an isentropic equation of state), and also due to the fact that left and right models in [1] are tightly connected in the long-time limit, even when gravity terms are accounted for, no straightforward comparison of results can be made between the present approach and the one introduced in [1]. However, owing to the well-balancing of source terms in the numerical algorithm of [1], one may presume that the latter approach is more accurate than the one introduced herein. Further analysis is still necessary to confirm this. Eventually, we would like to mention that both techniques may also be used in order to couple a standard two-fluid model, that assumes an instantaneous

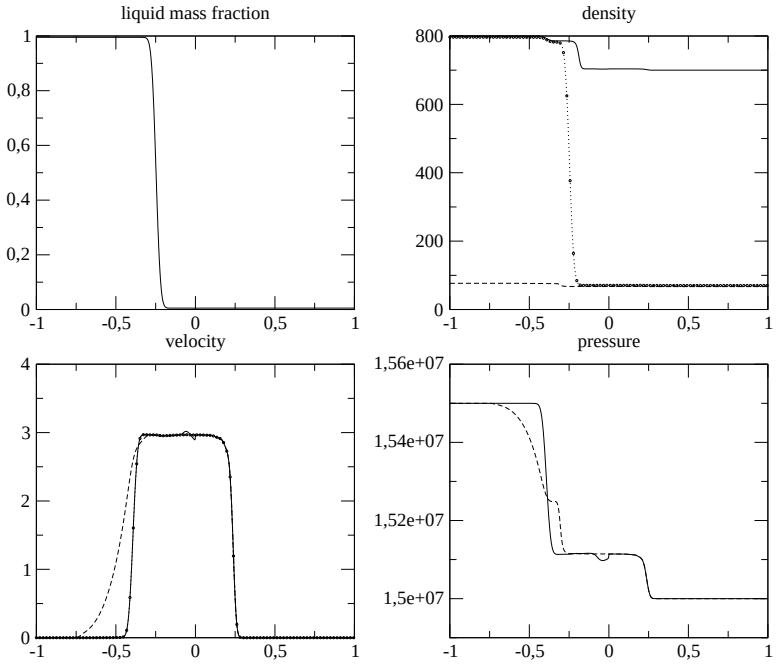

FIGURE 3. Same test case as in the previous figure, but the mesh now contains 2000 regular cells.

pressure equilibrium between both phases, with the homogeneous relaxation model HRM, as pointed out in [25].

4. **Comments and perspectives.** We have discussed in this paper a few coupling experiences involving distinct fluid models in unsteady situations. *Obviously, and mainly due to the number of studies, this paper only provides an overview of the methods that have been introduced, and the coupling methods have been described in a rather sketchy way. Thus we encourage readers that are interested in details to refer to appropriate references.* The main purpose in each case was to provide suitable boundary conditions through a fixed coupling interface. All models examined here issue from the two-phase flow literature, and involve highly non-linear and unsteady features. The basic methodology that has been applied in this work was to provide a *father model*, which implicitely enforces the definition of boundary conditions through the coupling interface. In each case, at least two fundamental questions arise:

- (i) Given a set of prescribed boundary conditions through the coupling interface, can we handle the numerical coupling of both models, and provide efficient numerical tools and practical recommendations for engineers?
- (ii) Is the set of boundary conditions meaningful from a physical/mathematical point of view?

Obviously, we have essentially focussed up to now on the first point, which of course is easier to examine than the second one. This has been briefly presented in the present paper, but thorough details are available in associated references. Nonetheless, we also tried to examine the second point in a few configurations,

among which we would like to quote the case of the coupling of models in free and porous media (see section 2.2). This one suggests that nice mathematical tools, which lead to reasonable numerical tools, might be improved in such a way that a better agreement with the physics are attained. We would also like to emphasize that one great difficulty dwells in the validation of numerical coupling techniques, due to the lack of analytical or reference solutions in many coupled situations.

Hence, the present work represents a humble contribution to the domain, and it essentially aims at providing numerical tools for engineers, rather than investigating the topic theoretically. Moreover, problems arising when including source terms $S(W)$ but also possible -even small- viscous contributions in left and right models have been disregarded. This represents another challenge for further applications and theoretical investigations. Part of our present work also consists in integrating tools in existing codes, which is no that obvious, mainly due to the fact that these existing codes have not been built with the perspective of subsequent coupling.

Acknowledgments. Part of the work of authors has been achieved within the framework of the NEPTUNE project, supported by CEA (Commissariat à l'Energie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP. The work has benefitted from very fruitful and useful discussions with Annalisa Ambroso (CEA), Christophe Chalons, Frédéric Coquel, Edwige Godlewski, Pierre-Arnaud Raviart and Nicolas Seguin (Laboratoire Jacques-Louis Lions, Université Pierre et Marie Curie, see references [2, 3, 4, 11, 13, 17, 18, 19] among others) and Thierry Gallouët (Université Aix-Marseille I). Eventually, we would like to acknowledge project leaders Frédéric Archambeau, Antoine Guelfi and also Olivier Marchand (EDF) for their kind support and encouragements. All computational facilities were provided by EDF.

REFERENCES

[1] A. Ambroso, C. Chalons, F. Coquel and T. Galié, *Relaxation and numerical approximation of a two-fluid two-pressure diphasic model*, Mathematical Modelling and Numerical Analysis, **43** (2009), 1063–1097.

[2] A. Ambroso, C. Chalons, F. Coquel, E. Godlewski, F. Lagoutiere, P. A. Raviart and N. Seguin, *The coupling of homogeneous models for two-phase flows*, International Journal of Finite Volumes, (electronic: http://www.latp.univ-mrs.fr/IJFV/), **4** (2007), 1–39.

[3] A. Ambroso, C. Chalons, F. Coquel, E. Godlewski, F. Lagoutiere, P. A. Raviart and N. Seguin, *Relaxation methods and coupling procedures*, International Journal for Numerical Methods in Fluids, **56** (2008), 1123–1129.

[4] A. Ambroso, C. Chalons, F. Coquel, E. Godlewski, F. Lagoutiere, P. A. Raviart and N. Seguin, *Coupling of general Lagrangian systems*, Mathematics of Computation, **77** (2008), 909–941.

[5] A. Ambroso, J. M. Hérard and O. Hurisse, *A method to couple HEM and HRM two-phase flow models*, Computers and Fluids, **38** (2009), 738–756.

[6] F. Bachmann, "Equations Hyperboliques Scalaires à Coefficients Discontinus," Ph.D thesis, Aix-Marseille I University, Marseille, France, 2005.

[7] M. R. Baer and J. W. Nunziato, *A two phase mixture theory for the deflagration to detonation (DDT) transition in reactive granular materials*, International Journal of Multiphase Flows, **12** (1986), 861–889.

[8] T. Buffard, T. Gallouët and J. M. Hérard, *A sequel to a rough Godunov scheme. Application to real gases*, Computers and Fluids, **29** (2000), 813–847.

[9] F. Caetano, "Sur Certains Problèmes de Linéarisation et de Couplage pour les Systèmes Hyperboliques Non-linéaires," Ph.D thesis, UPMC, Pierre et Marie Curie University, Paris, France, December 12, 2007.

[10] C. Cancès, "Ecoulements Diphasiques en Milieu Poreux Hétérogène: Modélisation et Analyse des Effets Liés aux Discontinuités de la Pression Capillaire," Ph.D thesis, Aix-Marseille I University, Marseille, France, 2008.
[11] C. Chalons, P. A. Raviart and N. Seguin, *The interfacial coupling of the gas dynamics equations*, Quart. Appl. Math., **66**(4) (2008), 659–705.
[12] F. Coquel, T. Gallouët, J. M. Hérard and N. Seguin, *Closure laws for two-fluid two-pressure models*, Comptes Rendus Mathématique, **I-334** (2002), 927–932.
[13] T. Galié, "Couplage Interfacial de Modèles en Mécanique des Fluides. Application aux Écoulements Diphasiques," Ph.D thesis, Pierre et Marie Curie University, Paris, France, March 31, 2009.
[14] T. Gallouët, J. M. Hérard and N. Seguin, *Numerical modelling of two-phase flows using the two-fluid two-pressure approach*, Mathematical Models and Methods in Applied Sciences, **14** (2004), 663–700.
[15] L. Girault and J. M. Hérard, *A two-fluid hyperbolic model in a porous medium*, Mathematical Modelling and Numerical Analysis, **in press** (2010).
[16] L. Girault and J. M. Hérard, *Multidimensional computations of a two-fluid hyperbolic model in a porous medium*, AIAA paper 2009-3888, http://www.aiaa.org/ (2009).
[17] E. Godlewski, *Coupling fluid models. Exploring some features of interfacial coupling*, in "Finite Volumes for Complex Applications V" (eds. R. Eymard and J.M. Hérard), ISTE-Wiley, (2008), 87–102.
[18] E. Godlewski and P. A. Raviart, *The numerical interface coupling of non-linear hyperbolic systems of conservation laws. I The scalar case*, Numer. Math., **97** (2004), 81–130.
[19] E. Godlewski, K. C. Le Thanh and P. A. Raviart, *The numerical interface coupling of non-linear hyperbolic systems of conservation laws. II the case of systems*, Mathematical Modelling and Numerical Analysis, **39** (2005), 649–692.
[20] J. M. Greenberg and A. Y. Leroux, *A well-balanced scheme for the numerical processing of source terms in hyperbolic equations*, SIAM Journal of Numerical Analysis, **33** (1996), 1–16.
[21] A. Guelfi, D. Bestion, M. Boucker, P. Boudier, P. Fillion, M. Grandotto, J. M. Hérard, E. Hervieu and P. Péturaud, *NEPTUNE: A new softaware platform for advanced nuclear thermal hydraulics*, Nuclear Science and Engineering, **156** (2007), 281–324.
[22] P. Helluy, J. M. Hérard and H. Mathis, *A well-balanced approximate Riemann solver for variable cross-section compressible flows*, AIAA paper 2009-3540, http://www.aiaa.org/ (2009).
[23] J. M. Hérard, *A rough scheme to couple free and porous media*, International Journal of Finite Volumes, (electronic: http://www.latp.univ-mrs.fr/IJFV/), **3(2)** (2006), 1–28.
[24] J. M. Hérard, *Un modèle hyperbolique diphasique bifluide en milieu poreux*, Comptes rendus Mécanique, **336** (2008), 650–655.
[25] J. M. Hérard and O. Hurisse, *Couplage interfacial d'un modèle homogène et d'un modèle bifluide*, EDF report H-I81-2006-04691-FR, **in French**, (2006).
[26] J. M. Hérard and O. Hurisse, *Coupling two and one-dimensional unsteady Euler equations through a thin interface*, Computers and Fluids, **36** (2007), 651–666.
[27] J. M. Hérard and O. Hurisse, *Boundary conditions for the coupling of two-phase flow models*, AIAA paper 2007-4458, http://www.aiaa.org/ (2007).
[28] J. M. Hérard and O. Hurisse, *Some recent numerical advances for two-phase flow modelling in NEPTUNE project*, International Journal for Numerical Methods in Fluids, **59** (2009), 285–307.
[29] D. Kröner and M. D. Thanh, *Numerical solutions to compressible flows in a nozzle with variable cross-section*, SIAM Journal of Numerical Analysis, **43** (2005), 796–824.
[30] J. Vovelle and N. Seguin, *Analysis and approximation of a scalar conservation law with a flux function with discontinuous coefficients*, Mathematical Models and Methods in Applied Sciences, **13** (2003), 221–257.

Received January 2010; revised June 2010.

E-mail address: jean-marc.herard@edf.fr
E-mail address: olivier.hurisse@edf.fr