Some open numerical problems arising in NEPTUNE project

Jean-Marc Hérard

Some open numerical problems in NEPTUNE project

Jean-Marc Hérard[⋆], †

[⋆] *EDF-Recherche Développement, Dept. MFTT, 6 quai Watier, 78400 Chatou, France*
† *CMI-LATP, UMR CNRS 6632, 39 rue Joliot Curie, 13453 Marseille, France*
Email: Jean-Marc.Herard@edf.fr, herard@cmi.univ-mrs.fr

ABSTRACT. *We discuss herein some open problems arising in the NEPTUNE project, which aims at preparing a new generation of two-phase flow codes covering the whole range of modelling scales. We focus on modelling topics and numerical methods. We first briefly list the main fields of associated investigations. We then concentrate on two specific aspects: the modelling of multiphase flows, and the numerical interfacial coupling of codes. Some recent achievements and a few open problems are discussed in the manuscript.*

RÉSUMÉ. *On examine ici quelques problèmes apparaissant dans le projet NEPTUNE, dont l'objectif est de préparer une nouvelle génération de codes diphasiques couvrant l'ensemble des échelles. On se concentre sur les aspects associés à la modélisation et aux méthodes numériques. On liste rapidement l'ensemble des thèmes abordés. On se focalise ensuite sur la modélisation des écoulements diphasiques et sur le couplage interfacial de codes. Quelques résultats récents et problèmes ouverts sont décrits.*

KEYWORDS: *Finite Volume Schemes / Multi-phase Flows / Coupling Methods*

MOTS-CLÉS : *Volumes Finis / Ecoulements multiphasiques / Techniques de couplage*

1. Introduction

The NEPTUNE Project gathers some of the efforts of CEA and EDF in terms of Research and Development programs (see [GBH05]). It aims at building the next generation of water-vapor two-phase flow codes for the nuclear energy applications. It should allow real time simulation using the system scale, but also perform multi-scale 3D computations. Three main axes arise. A first one is dedicated to the software development. A second one benefits from physical and numerical research investigations, while the third one is devoted to two-phase flow metrology, physical experiments and advanced instrumentation techniques ([PeH04]). Numerical methods are part of this

long-term research program, and should not only support developers, but also provide new ideas and methods to improve our understanding of multi-phase flows. Though Finite Volume methods ([EGH01]) are the keystone of almost all NEPTUNE developments, some methods rely on the use of Finite Element approach, or alternatively on the FVE approach. Hence, we will give first a very brief overview of some of recent achievements. We then will focus in section 2 and 3 on some topics connected with the modelling of multiphase flow, and with the coupling of codes.

We may basically say that some of the problems concern the establishment of sets of PDE which may account in an expected meaningful and relevant way for two-phase flow; the second problem concerns the numerical simulation of the latter systems.

In the first category, we wish to mention the original work on the modelling of interfaces reported in [ACK02], [CCJO4], [GuM04], and [Car04]. This not only examines the problem of the handling of pure convective effects, but it also investigates the closure and the approximation of interfacial mass transfer. A second contribution pertains to standard two-fluid models, and gives emphasis on the numerical strategies which may provide Finite Volume approximations on unstructured meshes (see [GKL01], [Kum 04], [Ndj06]).

The second category includes indeed very distinct contributions. Some are devoted to the numerical approximation of homogeneous models. Owing to the fact that we deal with nuclear applications in PWR essentially, there is first a need for preconditioning techniques for low Mach number applications, especially when dealing with upwinding conservative schemes (see [GuM03]). Moreover, water-vapor tabulations require that one may account for complex EOS (Equation Of State) in homogeneous codes, and this is precisely what has motivated the work ([GHS03]), which suggests some unified framework to deal with complex EOS. The "standard" problem of preconditioning also needs to be addressed, especially for the GENEPI code and other codes which rely on the classical projection techniques (see [BeG03a], [BeG03b]). Another important contribution ([For05]), which addresses the capabilities of Finite Volume Element techniques to handle accurate approximations of two-fluid models, is based on the early contribution [EmH02] which focused on Navier Stokes equations. The last research directions pertain to the use of Fictitious Domain methods, which might represent an alternative accurate way to account for complex geometries, on the basis of simple structured codes (see [ABF99], [Bel03], [RBA05a], [RBA05b]), and also to the improvement of unstructured Finite Volume codes ([Per01]).

2. A few open topics in the modelling of multi-phase flows

2.1. *A Hybrid Alternative Tool for Two-Fluid Models*

One of the main problems arising in the numerical modelling of two-phase flows is that sets of PDE which are expected to represent main patterns of such flows are still not clearly identified, or may suffer from severe drawbacks. This is probably particularly clear when focusing on the so-called two-fluid approach. Associated six-equation single-pressure models account for mass conservation, momentum and total phase energy balance within each phase. Everyone is aware of the potential loss of hyperbolicity of standard single-pressure two-fluid models in some areas, which unfortunately may easily arise, even for (expected so) simple sets of initial values, such as those described in [CEG98]. This main drawback seems to be closely linked with the assumption of local instantaneous pressure equilibrium. It seems that the blow up of codes may be quite easily postponed when applying for upwinding techniques, and restricting to coarse meshes. Nonetheless, it may be easily checked that the inner stabilization of the latter schemes is no longer sufficient on very fine meshes (with up to one million nodes in a 1D framework, see [HeH04] for instance), even when drag effects are accounted for.

This led some workers to reexamine "father" models known as two-fluid two-pressure models, following the pioneering work of [BaN86], [KSB97]. These benefit from classical properties, in the sense that: (i) the convective subset is hyperbolic for any physically relevant choice of the state variable; (ii) the whole set may be controlled by an entropy inequality for regular solutions, which meets agreement with true source terms and viscous terms. Under the constraint: $\alpha_1 + \alpha_2 = 1$, the basic form of the governing set of equations is the following:

$$\frac{\partial \alpha_k}{\partial t} + U_I \frac{\partial \alpha_k}{\partial x} = \lambda (P_k - P_{3-k})(P_k + P_{3-k})^{-1} \tag{1}$$

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \frac{\partial \alpha_k \rho_k U_k}{\partial x} = 0 \tag{2}$$

$$\frac{\partial \alpha_k \rho_k U_k}{\partial t} + \frac{\partial \alpha_k (\rho_k (U_k)^2 + P_k)}{\partial x} - P_I \frac{\partial \alpha_k}{\partial x} = -\mu (m_k m_{3-k})^{1/2} (U_k - U_{3-k}) \tag{3}$$

$$\frac{\partial (\alpha_k E_k)}{\partial t} + \frac{\partial (\alpha_k U_k (E_k + P_k))}{\partial x} + P_I \frac{\partial \alpha_k}{\partial t} = -\mu (m_k m_{3-k})^{1/2} U_I (U_k - U_{3-k}) \tag{4}$$

Where m_k, α_k, ρ_k, U_k, $E_k = \rho_k e_k + \rho_k U_k^2/2$, $e_k(\rho_k, Pk)$ and P_k respectively stand for the partial mass, the void fraction, the mean density, the mean velocity, the

total energy, the internal energy and the local pressure within phase k (for k=1,2). The problem of the closure of non-conservative products may even be circumvented for some specific choices of the interfacial velocity U_I and of the interfacial pressure P_I (see [GHS04]). Turbulence closures may be included, without breaking the keen wave structure (see for instance [GaS02], [Gav04] [Her05]). We however emphasize that the problem of existence and uniqueness of the solution of the 1D Riemann problem is still an open topic, mainly due to the great complexity of the wave structure, and to the possible resonance phenomenon, which seems to forbid uniqueness of solutions, unless some consistent criteria is added, which remains to be found... Straightforward upwinding techniques can be easily implemented, which provide reasonable numerical results. Due to the (six or) seven distinct eigenvalues, some of them being close to one another, intermediate states in the solution of the 1D Riemann problem can hardly be distinguished in some situations, unless one considers a huge mesh refinement (which can hardly be afforded when turning to 3D flow simulations!).

Another problem immediately occurs in this framework: is there some way to deal with both standard single- pressure- two-fluid models and those two-pressure hyperbolic models ? Based on recent work pertaining to relaxation methods, the temptation is great to take advantage of the father-son structure of the couple of models. Following [CoP98], [BBC02], [CCJ04], [Hel05], one may for instance compute the six-equation single pressure model, considering two steps as follows: (i) a first evolution step, which computes some approximation of solutions of the IVP problem connected with the hyperbolic seven-equation two-pressure- model, followed by: (ii) an instantaneous relaxation step which locally equilibrates both pressure fields at the end of each time step. Obviously, the input of (i) is the output of (ii), which guarantees pressure equilibrium. The HAT (Hybrid Alternative Tool) algorithm enjoys rather interesting properties. The whole scheme identifies with the one in [GHS04], when the pressure relaxation time remains bounded. The spirit is almost the same as the one from [CEG98], and we also underline that the numerical treatment of step (ii) is exactly the same, and provides consistent approximations of the mean pressure field. It might represent some possible way to tune models through the next coming years (see [Nus07]). It also seems an appealing way to provide meaningful boundary conditions in six-equation two-fluid models.

Results: We show in [Her05] some approximations of the solution of a Riemann problem using a turbulent closure within each phase. The structure of the density fields, the velocity fields, and the modified pressure fields ($\pi_k = P_k + 2K_k/3$) is displayed. One may also refer to [Her05] which examines from a practical point of view the influence of the couple (U_I, P_I) on the computational results on very fine grids (and thus the true influence of this closure). This is quite interesting due to the fact that some authors might prefer using other closures.

Figure 1 shows the behavior of the void fraction α_k when computing approximate solutions of a standard two-fluid model on a coarse mesh and on very fine meshes (250000 cells). This latter result issues from [HeH04].

2.2. *Three-field Models*

Some specific applications in the nuclear energy require considering a mixture of liquid droplets in a continuum of vapor surrounded by a continuous liquid phase. The expected velocity of droplets inside the gas phase is clearly much higher than the local gas velocity. This is referred to as a three-field pattern in the nuclear community, and emerging ideas seem to retain an alternative way to tackle this problem, which consists in the modelling of three-phase flows. We may choose this approach, but a straightforward consequence is that the potential lack of hyperbolicity obviously once more arises. An obvious idea is to mimic the two-fluid approach discussed above, and thus constructing hyperbolic three-phase models. A first attempt is sketched in [Her04b]. Among others, the associated results suggest that the counterpart of the closure:

$$U_I = \frac{(m_1 U_1 + m_2 U_2)}{(m_1 + m_2)} \tag{5}$$

(in the two-fluid two-pressure approach), which indeed makes sense in this particular framework (see [GHS04]), no longer seems suitable for three-phase flows. Moreover, admissible forms (that is forms which are consistent with the overall entropy inequality) of interfacial mass and momentum transfer terms slightly differ from their two-phase counterpart. It nonetheless once again provides a hyperbolic framework to tackle three-phase flows, and results seem to confirm that the resonance phenomena is the only barrier remaining before solving the one dimensional Riemann problem. This first attempt obviously requires deeper investigations.

This approach can also be useful to compute approximations of single-pressure models with help of relaxation techniques, at least when equations of state are simple enough.

Results: We show on figure 2 the structure of the pressure fields when computing a Riemann problem, while neglecting drag terms. Once more, the great number of intermediate states urges the need for very accurate solvers on rather coarse meshes.

3. Unsteady coupling of codes through a thin interface

There is nowadays a true need for coupling techniques in order to cope with industrial applications using current codes. This occurs for instance when computing the

whole primary coolant circuit of the pressurized power reactor with help of different codes, for instance 3D codes (such as FLICA IV, THYC, ...), which rely on the homogeneous approach to describe the core, and 1D codes (such as CATHARE, ...), which apply for the standard two-fluid approach to describe patterns in pipes. No tools have been prepared for that purpose over the last years, and some simple -even steady- simulations which require some coupling have already exhibited major deficiencies. The NEPTUNE project team has decided to give special emphasis on that topic, in close collaboration with the members of the working group [ACC03]. It is also expected that this work will also contribute to extend our understanding of sole systems.

The main goal is to cope with the interfacial coupling of the system scale, the component scale and the local 3D CFD scale. The basic strategy up to now consists in decoupling all effects, and then focusing on specific problems. We refer to [Amb04] which synthesizes the needs, basic ideas and elementary tools available from the literature, whenever one considers the scalar case or the system case (see [GoR04], [GLR05]). We rather quickly detail below some of the recent achievements in that work package, and some ongoing work with Olivier Hurisse (see sections 3.1 to 3.3, and 3.5). The reader is invited to read [ACC04] and [ACC05] which investigate topics 3.4 and 3.5 .

3.1. *Well balanced schemes versus fractional step methods.*

Multiphase CFD codes provide approximations of PDE which basically rely on contributions which account for convective effects and source terms (mass, energy and momentum interfacial transfer). The coupling of two codes will thus involve two sets of PDEs with different time scales (time scales associated with flashing phenomena, condensation or drag effects are indeed quite different!). Even before achieving any coupling, one may obviously wonder how numerical methods will deal with the whole. Thus an obvious question is: what is the true accuracy of "standard" discretizations when aiming at computing hyperbolic systems with source terms, especially when highly unsteady patterns travel through the interface between these codes. An elementary investigation has been performed, considering the same set of equations on both sides of the interface, while focusing on two rather classical classes: (i) fractional step methods, which treat separately convection and sources, and (ii) well-balanced schemes ([GrL96]) which have been designed to get accurate approximations of steady states on coarse meshes. This work is discussed in detail in [GHH05], where the model problem is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U)}{\partial x} = 0 \tag{6}$$

$$\frac{\partial \rho x}{\partial t} + \frac{\partial (\rho x U)}{\partial x} = \rho S(\rho, x, \tau) \tag{7}$$

$$\frac{\partial \rho U}{\partial t} + \frac{\partial (\rho U^2 + P(\rho))}{\partial x} = 0 \tag{8}$$

Where ρ, U, x respectively stand for the mean density, the mean velocity of the mixture of water and vapor, and the vapor concentration, and $P'(\rho) > 0$. The simple form of the source term $S(\rho, x, \tau)$ which contains a time scale τ which may be tuned, enables to carry on analytical work. In some situations, both explicit schemes compare quite well, but when the time scale τ becomes too small, results are much in favor of the fractional step approach, even if the latter does not seem to be the ultimate approach.

Results: We show on figure 3 some computational results which correspond to the propagation of a shock wave through the interface ($x = 0$). It clearly arises here that for this magnitude of τ, the FSM performs better than the WBS. We emphasize that the low Mach number regions contribute to the loss of accuracy, since $h/(U\tau) = M^{-1}(1 + M)(\delta t/\tau)(CFL)^{-1}$ is not always small when compared with 1 (where h and δt refer to the mean mesh size and the time step respectively, and assuming CFL is close to 1).

3.2. *Free medium / porous medium*

A clearly identified problem occurs when the flow of fluid in a free medium enters a porous region (the porosity will be denoted ϵ). We must assume that the porous formulation (on the right side for instance) has been fixed. We start first with the set of PDE which corresponds to the isentropic Euler equations in a regular porous region. The flow on the left hand side is also assumed to be governed by conservative- Euler equations. The problem now is to provide a suitable exchange of information through the interface. One may simply suggest that a meaningful interface model simply is the one on the RHS:

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial (\rho \epsilon U)}{\partial x} = 0 \tag{9}$$

$$\frac{\partial \rho \epsilon U}{\partial t} + \frac{\partial (\rho \epsilon U^2)}{\partial x} + \epsilon \frac{\partial P(\rho)}{\partial x} = 0 \tag{10}$$

Since the latter enables to retrieve the set on the LHS. The whole also guarantees that the mean mass flow rate ($\epsilon \rho U$) is continuous through the interface, which agrees with physical requirements. The continuity of the second Riemann invariant of the stationary wave is nonetheless much more conjectural. Actually the entropy inequality only

suggests that the sign of its variation should be correlated with the sign of the mass flow rate through the interface. Moreover, a drawback immediately appears, since one expects that physical grounds should guide the variation of this second invariant. Eventually, one may wonder whether Riemann solvers will handle the whole process, and what the influence of the path that connects ϵ on both sides of the interface is.

Some attempts to handle this problem are described in [Her04a]. This work tries to address some of the questions above. It obviously arises that almost any interface and cell scheme will correctly treat small variations of ϵ, but clearly not ratios of 1 to .05. This (crude?) approach, which is basically influenced by underlying ideas from [GrL96], provides rather satisfactory results from an engineering point of view, but it still requires improvements. A reasonable requirement is that one should at least be able to prescribe some loss of momentum through the interface (this is currently investigated by the working group [ACC03]).

Results: Figure 4 shows some shock wave coming from the free medium and propagating to the right side through the interface located on ($x = 0.5$). Cell values of the mean flow rate through the interface are correctly represented. The jump of the second invariant (see figure 4) through the interface is in agreement with the overall entropy inequality. Moreover, the approximate solution -on sufficiently fine meshes- does not depend on the interface path, whatever one introduces a discontinuous path, or a linear / quadratic path -on a given number of cells- to "smooth" the interface separating the two codes.

3.3. *Flows through pipes and reactors*

When trying to couple the CATHARE code with the FLICA IV code, one is urged to handle the information between a 3D code and a one-dimensional code. In a first step, one may for instance consider the isentropic Euler set of equations to examine this -since the total energy equation does not introduce any new eigenvalue in the convective problem in a multi-dimensional framework-. One may even restrict to a 2D framework, since the 3D case does not introduce any further difficulty, and would constrain much the capacities for mesh refinement. Hence the set of PDEs on the right hand side will be:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U)}{\partial x} + \frac{\partial (\rho V)}{\partial y} = 0 \tag{11}$$

$$\frac{\partial \rho U}{\partial t} + \frac{\partial (\rho U^2 + P(\rho))}{\partial x} + \frac{\partial (\rho U V)}{\partial y} = 0 \tag{12}$$

$$\frac{\partial \rho V}{\partial t} + \frac{\partial (\rho U V)}{\partial x} + \frac{\partial (\rho V^2 + P(\rho))}{\partial y} = 0 \tag{13}$$

And there will be its straightforward 1D counterpart on the left hand side, setting $V = 0$. We also underline that the introduction of a variable cross section in the pipe has no influence on the following results. It first quickly arises that the projection along the x-axis of the $2D$ model on the RHS is not a suitable candidate at the interface, since it does not ensure that the expected and necessary - constraint $V((x - x_{int})/t = 0^-) = 0$ holds. Nevertheless, the interface model ([HeH05]):

$$\frac{\partial Z}{\partial t} = 0 \tag{14}$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U_n)}{\partial n} = 0 \tag{15}$$

$$\frac{\partial \rho U_n}{\partial t} + \frac{\partial (\rho U_n^2 + P(\rho))}{\partial n} = 0 \tag{16}$$

$$\frac{\partial U_\tau}{\partial t} + Z U_n \frac{\partial U_\tau}{\partial n} = 0 \tag{17}$$

(with $n = x$, $U_n = U.n$ and $U_\tau = U.\tau$) obeys the different constraints. One may thus derive a Godunov scheme which is in agreement with $V((x-xint)/t = 0^-) = 0$. An important feature here is that we may face the approximate solutions to the true solution that may be obtained while using the 2D code everywhere and refining much the mesh size. Other points which are addressed in this paper concern the influence of the EOS (water or vapor), the influence of the scheme at the interface, the major role of the interface location in the pipe, and of course the influence of the mesh size. Computations try to maximize the mass flow rate at the interface, which is combined with great values of tangential velocity, in order to maximize the "pollution" around the interface.

Results: The next figure (fig. 5) shows the distribution of transverse momentum in a pipe and a tank, using a 1D code in the pipe and a 2D code in the tank. Approximate Godunov solvers are used in both codes to discretize shallow water equations. The results show rather fair agreement with results associated with a full $2D$ code, in terms of L^1 norm and L^∞ norm of the error. Obviously, the 1D code inhibits the convergence to the true solution when the mesh is highly refined.

3.4. *The influence of thermodynamical EOS*

The impact of discrepancies in EOS formulations or tabulations is also a major problem, since very small variations on thermodynamical coefficients may greatly change results. Actually almost all thermodynamical approaches implemented in industrial codes differ. A straightforward consequence is that the interfacial coupling of codes involving different EOS may pollute the whole solution. The behavior of expected similar EOS with arbitrary (assumed null!) jumps of coefficients, is currently examined and reported in [ACC04], where authors take advantage of relaxation tech-

niques to avoid the resonance phenomena which may occur when applying brutally for basic ideas issuing from "upwinding of source terms in steady situations". It is strongly connected with the next issue.

3.5. *Relaxed models / Unrelaxed models*

In a following step, we need to investigate the coupling of almost similar models in a one-dimensional open medium framework, which typically will correspond to the coupling of a Homogeneous Relaxation Model and a Homogeneous Equilibrium Model (namely HRM and HEM). This interconnects with the former section 3.1. Some recent details on that topic will be presented in [ACC05], [Hur06]. Once again it suggests that the use of relaxation methods may offer a comfortable framework to derive coupling techniques.

3.6. *Homogeneous models / Two-fluid models*

The main goal of such concern is of course the clear definition of the information to be exchanged between a six- equation two-fluid model and a four-equation homogeneous model.

4. Other topics and open questions

We have clearly completely omitted talking about the direct numerical simulation of two-phase flows herein. This actually fully lies within the scope of the NEPTUNE project, and we refer to the paper [BeG04] that discusses that topic in detail. Obviously, the growing capacities of computers during the last ten years suggest that one should try to define closure laws with help of computer results, rather than using some mixed mathematical and phenomenological approaches. It may be argued too that the last forty-year experience in the framework of single-phase turbulence should be accounted for. One question is to define whether we should reproduce the counterpart of the latter approach over the next ten years, or whether we should consider an alternative way gathering several approaches. For instance, hybrid modelling has been totally disregarded, which might represent some possible weakness. The temptation is great to shift from classical Adaptative Mesh Refinement to Adaptative Model Refinement. This however implicitly means that the relaxation process which enables to tune locally from model M1 to one of its relaxed forms is totally under control, which is not the case up to now! It also questions whether one should use CPU to refine meshes on coarse models, or alternatively to refine models on coarse meshes. As a matter of fact, no specific approach has been privileged until now, which enables to benefit from experience of various workers in different fields, and does not inhibit new possibly improving- developments. The DNS approach represents such an alternative,

and should not be disregarded.

Another point that deserves interest concerns the use of relaxation techniques. The increasing gain of interest in relaxation methods essentially seems motivated by pragmatism. One of the main interests of these methods is that they enable to give some kind of uniform frame to tackle with complex problems. When dealing with the simulation of Euler equations with any kind of EOS, they allow using simple Riemann solvers based on simplified EOS, and meanwhile ensure the convergence towards the right weak solutions, even if the real EOS is complex. When turning to the prediction of two phase unsteady flows with the help of two-fluid models, they enable to uniformize the algorithm, and suggest some hybrid way of modelling: standard two-fluid models on coarse meshes, and two-fluid two pressure models on fine meshes. This however implicitly means that an extensive use of computers will become a routine in the near future. Moreover, it might once more re-open the wide debate on the need for hyperbolicity of the first order differential subset of convective terms.

A third point concerns the true accuracy of current and future computations. If we focus on the last five years, it looks as if there were in fact a tendency to utilize computational resources in order to enlarge the size of the computational physical domain, or alternatively the number of non-linear governing equations. Both of these two tend to decrease the local accuracy, which is in some way rather amazing! A direct consequence is also that one should right now consider the hybridizing of models as a key point of the future, with deep and consistent connection with "standard" local mesh refinement. Some work on the latter topic has just begun.

Acknowledgments

The NEPTUNE project is mainly funded by CEA and EDF, with a complementary support by IRSN and FRAMATOME-ANP. Contributors to the work package on numerical methods are : Annalisa Ambroso, Michel Belliard, Philippe Emonot, Gauthier Fauchet, Marc Grandotto, Samuel Kokh, Anela Kumbaro, and Hervé Lemonnier, from CEA, and Stéphane Mimouni from EDF. The whole benefits from contributions of past and present PhD students : Florian Caro, Thomas Fortin, Vincent Guillemaud, Olivier Hurisse, Angelo Muronne, Isabelle Ramière, Jean-Michel Rovarch and Nicolas Seguin. Grégoire Allaire, Philippe Angot, Christophe Chalons, Frédéric Coquel, Thierry Gallouët, Edwige Godlewski, F. Lagoutière, Pierre-Arnaud Raviart and other members of the Paris VI-JLL working group [ACC03] are also acknowledged for their kind support and advice. Many thanks to the "bosses" too: Daniel Caruge, Bernard Faydide and Olivier Marchand.

The author is also indebted to Annalisa Ambroso, Thierry Gallouët, Antoine Guelfi (NEPTUNE project manager), Raphaèle Herbin and Olivier Marchand who kindly accepted to read the initial version of the manuscript, which helped much.

Appendix A

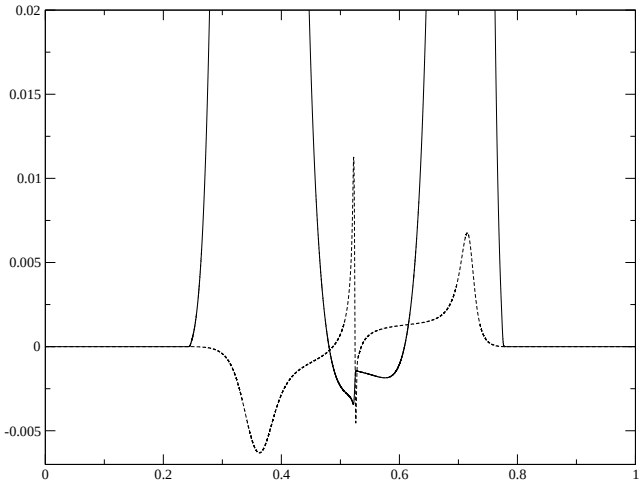

Figure 1. $A1$*: Relative velocity* U_r *and void fraction profiles when computing approximations of solutions, using the initial value problem given in [CEG 98]. On the basis of the two-pressure approach, the relaxation technique enables to compute approximations of the two-fluid single-pressure model which accounts for drag source terms. The bubble diameter is equal to* $d = 10^{-3}$*. The present mesh contains 250 000 cells.*

5. References

[ACK02] ALLAIRE, G., CLERC,S., AND KOKH, S. 2002. A five equation model for the simulation of interfaces between compressible fluids. *J. of Comp. Phys.*, 181, 577-616.

[Amb 04] AMBROSO, A. , 2004. An introduction to the problem of interfacial coupling of thermohydraulic models (unpublished).

[ACC03] AMBROSO, A. , CHALONS, C., COQUEL, F., GODLEWSKI, E., LAGOUTIÈRE, F., AND RAVIART, P.A., 2003. Joint CEA-SFME / Laboratoire Jacques-Louis Lions Working Group on the Coupling of Models in Thermal Hydraulics.

[ACC04] AMBROSO, A. , CHALONS, COQUEL, F., C., GODLEWSKI, E., LAGOUTIÈRE, F., RAVIART, P.A., SEGUIN, N., AND HÉRARD, J.M., 2005. Coupling of multiphase flow models. *To appear in the proceedings of NURETH11*. Aix en Provence, Popes' Palace.

[ACC05] AMBROSO, A. , CHALONS, C., COQUEL, F., GODLEWSKI, E., LAGOUTIÈRE, F., RAVIART, P.A., AND SEGUIN, N., 2005. Homogeneous models with phase transition: coupling by Finite Volume methods. *To appear in the proceedings of FVCA4*.

[ABF99] ANGOT, P. , BRUNEAU, C.H., AND FABRIE, P., 1999. A penalization method to take into account obstacle in incompressible viscous flows. *Numerische Mathematik*, 81, 497-520.

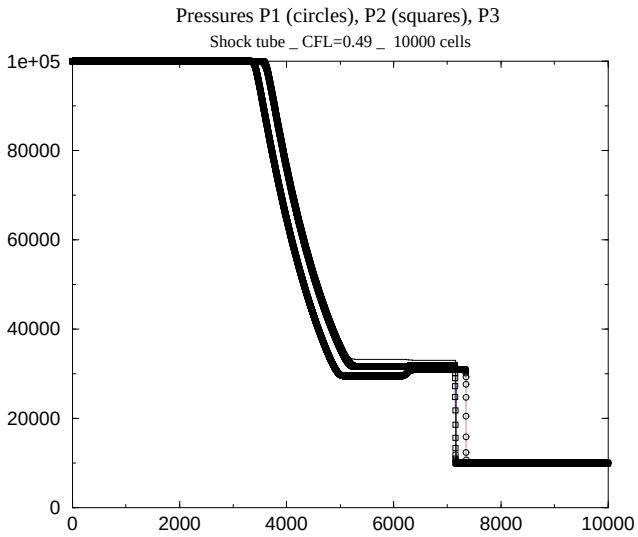

Figure 2. *A2: Behaviour of the three pressures when computing a standard Riemann problem (hyperbolic three-phase flow model). The three phases are at rest and share the same pressure on each side of the membrane before the beginning of the computation. The three pressures vary through the contact discontinuity which supports the void fraction disontinuity. The mesh contains 10000 cells. All source terms have been neglected here. A non conservative version of the Rusanov scheme has been used here.*

[ALR05] ANGOT, P., LOMENEDE, H., RAMIÈRE, I., 2005. A general fictitious domain method with non-conforming structured mesh. *Submitted for contribution to FVCA4*.

[BaN86] BAER, M.R., AND NUNZIATO, J.W., 1986. A two phase mixture theory for the deflagration to detonation transition (DDT) in reactive granular materials. *International Journal Multiphase Flow*, 12-6, 861-889.

[BaH04] BARBERON, T. AND HELLUY, P., 2004, Finite Volume simulation of cavitating flows. *Computers and Fluids*, vol. 35.

[BBC02] BAUDIN, N., BERTHON, C., COQUEL, F., HOCHE , P., MASSON, R., AND TRAN, H., 2002. A relaxation method for two phase flow models with hydrodynamic closure law, *preprint MAB 02-02*, Bordeaux I, France.

[Bel03] BELLIARD, M. , 2003. Using a fictitious domain approach for the numerical modelling of two phase flows in nuclear power plant components. *Proceedings of Supercomputing for Nuclear Applications*, SNA2003.

[BeG02] BELLIARD, M., AND GRANDOTTO, M., 2002. Computation of two phase flow in steam generator using domain decomposition method. *Nuclear Engineering and Design*, 213, 223-239.

[BeG03a] BELLIARD, M., AND GRANDOTTO, M., 2003a. Multigrid preconditioning of the steam generator two phase mixture balance equations in the GENEPI software. *Proceedings*

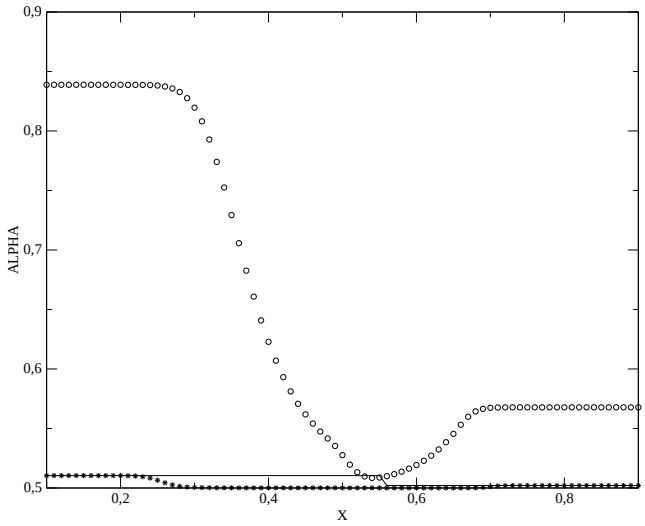

Figure 3. $B1$ *A comparison of the first Well Balanced Scheme (circles) and the second Well Balanced Scheme (stars) with the Fractional Step Method (FSM) . The mesh contains 100 cells. The relaxation time in the mass transfer term is* $\tau = 0.0001$. *The adimensional parameter* $\frac{h}{U\tau}$ *is ot small compared with* 1.

of NURETH10.

[BeG03b] BELLIARD, M., AND GRANDOTTO, M., 2003b. Local zoom computation of two phase flows in steam generators using a local defect correction method. *Numerical Heat and Mass Transfer*, 43, 1-5.

[BeG04] BESTION, D., AND GUELFI, A. 2004. Multiscale analysis of thermal hydraulics of nuclear reactors : the NEPTUNE project. *SHF meeting on Advances in the modelling methodologies of two-phase flows*, 24-26/11/2004, Lyon, France.

[CCJ04] CARO, F., COQUEL, F., JAMET, D , AND KOKH, S., 2004. DINMOD: a diffusive interface model for two phase flow modelling. *IRMA series in mathematics and theoretical physics*, to appear.

[Car04] CARO, F., 2004. PhD thesis, Modélisation et simulation numérique des transitions de phase liquide-vapeur, 24/11/04, Ecole Polytechnique, Paris, France.

[CEG98] COQUEL, F., EL AMINE, K., GODLEWSKI, E., RASCLE, P. AND PERTHAME, B., 1998. A numerical method using upwind schemes for the resolution of twophase flows *J. of Comp. Phys.*, 136, pp. 272-288.

[CoP98] COQUEL, F., AND PERTHAME, B., 1998. Relaxation of energy and approximate Riemann solvers for general pressure laws in fluid dynamics. *SIAM J. of Num. Analysis*, 35, pp. 2223-2249.

[EmH02] EMONOT, P., AND HEIB, S. 2002. Convergence of a new non conforming equal order Finite Volume Element method for the Stokes problem. *Journal of Sci. Computing.*

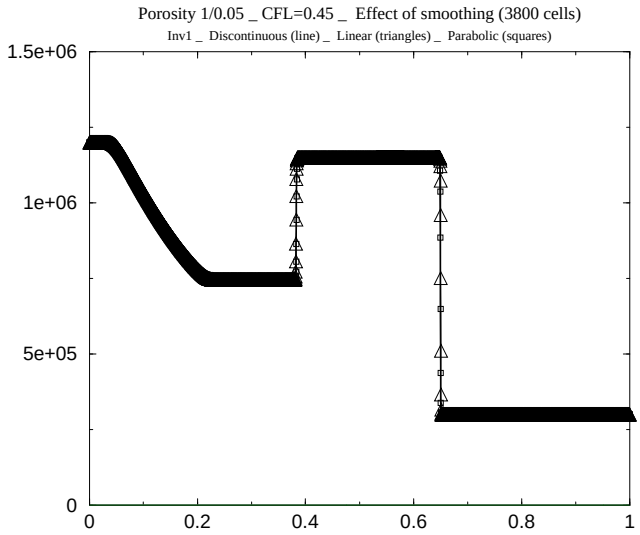

Figure 4. *$B2$: Second steady Riemann invariant $U^2/2 + \psi(\rho)$ around the interface between the two codes. The free and porous ($\epsilon = 0.05$) media respectively stand on the left and right sides of $x = 0.5$. A shock wave is coming from the left and hits this porous interface. Part of the incoming wave is reflected, and part is transmitted through the porous medium. The interface approximate Riemann solver, which relies on the VFRoe-ncv approach, enforces the continuity of both steady Riemann invariants. The numerical solution is in agreement with the entropy inequality.*

[EGH01] EYMARD, R., GALLOUËT, T., HERBIN R., 2001, Finite Volume Methods. *Handbook for numerical analysis*. P.G. Ciarlet and P.L.L Lions editors, vol. VII.

[For05] FORTIN, T., 2005. PhD thesis, in preparation.

[GHH05] GALLOUËT, T., HÉRARD, J.M., HURISSE, O. AND LE ROUX, A.Y., 2005. Well balanced schemes versus fractional step method for hyperbolic systems with source terms , *submitted for publication.*

[GHS03] GALLOUËT, T., HÉRARD, J.M. AND SEGUIN, N. 2003. A hybrid scheme to compute contact discontinuities in one dimensional Euler systems. *Mathematical Models and Numerical Analysis*, 36, 1133-1159.

[GHS04] GALLOUËT, T., HÉRARD, J.M. AND SEGUIN, N. 2004. Numerical modelling of two phase flows using the two-fluid two-pressure approach. *Mathematical Models and Methods in Applied Sciences*, 14, 663-700.

[Gav04] GAVRILYUK, S., 2005. Acoustic properties of a two-fluid compressible mixture with micro inertia. *Eur. J. Mechanics / B*, vol. 24, pp. 397-406.

[GaG99] GAVRILYUK, S., AND GOUIN, H., 1999. A new form of governing equations of fluids arising from Hamilton's principle. *Int. Journal of Eng. Sci.*, 37, 1495-1520.

[GGP97] GAVRILYUK, S., GOUIN, H., AND PEREPECHKO, Y.V., 1997. A variational princi-

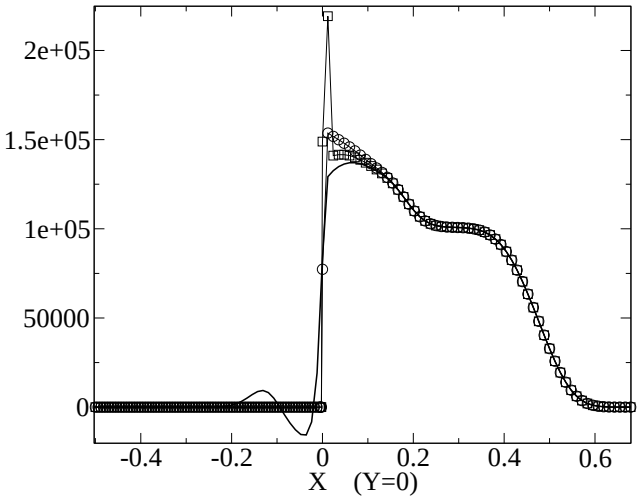

Figure 5. *$B3$: Mean y-momentum around the interface between the two codes, along the x-axis aligned with the pipe. The one-dimensional code and the two-dimensional code respectively stand on the left and right sides of $x = 0$. They both compute approximations of isentropic Euler equations. The initial condition generates a strong shock wave which propagates from the right to the left. Approximate RGodunov solvers are used to compute fluxes for standard interfaces. The whole mesh contains around 40000 cells. The comparison includes a computation with the 2D code used within the pipe (line), together with the non-conservative coupling (circles), and a rough conservative algorithm (squares).*

ple for two-fluid models. *CRAS Paris IIb.*, 324, 483-490.

[GaS02] GAVRILYUK, S., AND SAUREL, R., 2002. Mathematical and numerical modelling of two phase compressible flows with micro-inertia. *J. of Comp. Phys.*, 175, 326-360.

[GKL01] GHIDAGLIA, J.M., KUMBARO, A., AND LE COQ, G., 2001. On the numerical simulation to two fluid models via a cell center Finite Volume method. *European Journal of Mechanics / B Fluids*. 20.

[GoR04] GODLEWSKI, E. , AND RAVIART, P.A.. 2004. The numerical coupling of non-linear hyperbolic systems of conservation laws. I : The scalar case. *Numerische Mathematik*, 97, 81-130.

[GLR04] GODLEWSKI, E., LE TANH , K. AND RAVIART, P.A., 2004. The numerical coupling of non-linear hyperbolic systems of conservation laws. II : The system case, *to appear in M2AN*.

[GrL96] GREENBERG, J.M., AND LE ROUX, A.Y., 1996. A well balanced scheme for the numerical processing of source terms in hyperbolic equations. *SIAM J. of Num. Analysis*, 33-1 , 1-16.

[GBH05] GUELFI, A., BOUCKER, M., HÉRARD, J.M., PÉTURAUD, P., BESTION, D.,

BOUDIER, P., FILLION, P., GRANDOTTO, M., HERVIEU, E. , 2005. A new multi scale platform for advanced nuclear thermal hydraulics: status and perspectives of the NEPTUNE project, Contribution to NURETH11.

[GuM03] GUILLARD, H. AND MURONNE, A., 2003 A five equation reduced model for compressible two phase flow problems. INRIA report 4778, *to appear in J. of Comp. Phys.*.

[GuM04] GUILLARD, H. AND MURONNE, A., 2004. On the behaviour of upwind schemes in the low Mach number limit: II. Godunov type schemes. *Computers and Fluids*, 33, 655-675.

[Gui06] GUILLEMAUD, V., 2006. PhD thesis, Université de Provence, in preparation.

[Hel05] HELLUY, P., 2005, Simulation numérique des écoulements multiphasiques : de la théorie aux applications, 06/01/05, Toulon, http ://helluy.univ-tln.fr/ADMIN/habilitation.pdf

[Her04a] HÉRARD, J.M., 2004a. Naive schemes to compute isentropic flows between free and porous medium. EDF report HI-81/04/08/A (unpublished).

[Her04b] HÉRARD, J.M., 2004b. A simple hyperbolic model for multiphase flows (submitted)

[Her05] HÉRARD, J.M., 2005. A turbulent hyperbolic two-phase flow model (submitted)

[HeH04] HÉRARD, J.M., AND HURISSE, O. 2004. A relaxation method to compute two-fluid models. *EDF report HI-81/05/01/A.*

[HeH05] HÉRARD, J.M., AND HURISSE, O. 2005. Coupling one and two-dimensional models through a thin interface. *AIAA paper 2005-4718*, 17th AIAA CFD Conference, Toronto, Canada, 6-9 june 2005.

[Hur06] HURISSE, O., 2006. PhD thesis, Université Aix Marseille I, in preparation.

[KSB97] KAPILA, A.K., SON, S.F., BDZIL, J.B., MENIKOFF, R., AND STEWART, D.S., 1997. Two phase modelling of DDT: structure of the velocity relaxation zone. *Physics of Fluids*, 9-12, 3885-3897.

[Kum04] KUMBARO, A., 2004. Simulation of two-phase flows using a multi-size group model, *Third Int. Symposium on Two Phase Flow Modelling And Experiments*, Pisa, Italy.

[Ndj06] NDJINGA, M., 2006. PhD thesis, in preparation, Ecole Centrale de Paris , Paris, France.

[Nus07] NUSSBAUM, J., 2007. PhD thesis, Université de Toulon, in preparation.

[Per01] PERRON, S., 2001. Résolution de problèmes d'écoulements thermiques en 3D avec une nouvelle méthode de Volumes Finis, PhD thesis, 15/11/01, Université du Québec à Chicoutimi, Québec, Canada.

[PeH04] PÉTURAUD, O., AND HERVIEU, E., 2004. Physical validation issue of the NEPTUNEtwo-phase modelling : validation plan to be adopted, experimental programs to be set up and associated instrumentation techniques developed. *SHF meeting on Advances in the modelling methodologies of two-phase flows*, 24-26/11/2004, Lyon, France.

[RBA05a] RAMIÈRE, I., BELLIARD, M. AND ANGOT, P., 2005. On the simulation of nuclear power plant components using a fictitious domain approach. *Accepted for contribution to NURETH11*.

[RBA05b] RAMIÈRE, I., BELLIARD, M. AND ANGOT, P., 2005. Using a fictitious domain approach for solving a diffusion problem. Submitted

[Ram06] RAMIÈRE, I., 2006. PhD thesis, Université Aix Marseille II, in preparation.