177ème session du comité Scientifique et Technique de la
SOCIETE HYDROTECHNIQUE DE FRANCE
25 rue des Favorites - 75015 PARIS - Tél. : 01 42 50 91 03 -
Fax : 01 42 50 59 83 - mail : shf@shf.asso.fr - site www.shf.asso.fr

ADVANCES IN THE MODELING METHODOLOGIES OF TWO-PHASE FLOWS

Lyons, France, November 24-26, 2004

LISTE DES COMMUNICATIONS - *LIST OF PAPERS*

Pour structurer les sessions, on propose de classer les contributions selon les thèmes suivants.

PI : Projets industriels, exposé des besoins industriels suggérés par les applications.
I : Accès aux connaissances nouvelles par les techniques expérimentales et techniques de mesure associées
M : Modélisation physique
N : Développements de techniques numériques pour la modélisation
A : Applications de la modélisation des écoulements diphasiques pour les procédés industriels. Problèmes mal résolus et difficultés

Le colloque comportera 4 sessions formées par les contributions à ces thèmes selon l'ordre suivant: PI, I, M et N, A.

To collect the presentations into sessions, the following topics are proposed.

PI: Industrial projects, presentation of industrial needs derived from indutrial problems
I: How to get validation data from dedicated experiments and by developing measuring techniques
M: Physical modeling
N: Develoment of numerical techniques for solving equations derived from the models
A: CFD application to industrial problems, open problems and shortcomings

The coloquium will consist in 4 sessions collecting the contributions according to the following order: PI, I, M and N, A.

✔ ✔ **I** — **001**: Prasser, H.-M., *Wire-mesh sensors: an experimental tool for two-phase CFD model development and code validation* (Forshungzentrum Rossendorf, Germany)

✔ ✔ **I** — **002**: Prasser, H.-M., Hampel, U., *Gamma and X-Ray tomography for transient two-phase flows and other instrumentation developed by Rossendorf* (Forshungzentrum Rossendorf, Germany)

✔ ✕ **I** — **003**: Legoupil, S., *X-Ray imaging for fluid flow measurement: present and future trends* (CEA/Saclay, France)

✔ ✔ **N** — **004**: Wörner, M., Ghidersa, B. E., Ilic, M., Cacuci, D., G., *Volume-of-fluid method based numerical simulations of gas-liquid two-phase flows in confined geometries* (Forshungzentrum Karlsruhe, Germany)

✔ ✔ **M** — **005**: Kim, S., Ishii, M., *Dynamic modeling of interfacial structures via interfacial area transport equation* (Univ. of Missouri, Rolla, Purdue Univ., USA)

✔ ✔ **A** — **006**: Hager, W., H., Gisonni, C., *Supercritical flow in sewer manholes* (ETH Zürich, Switzerland, Napoli Univ., Italy)

✔ ✔ **N** — **007**: Banerjee, S., Badalassi, V., Dwivedi, V., Nave, J.-C., Hall, D., *The direct numerical simulation of two-phase flows with interface capturing methods* (UCSB, USA)

✔ ✔ **I** — **008**: Hassan, Y., *Drag reduction via microbubble injection in boundary layers of channel flows* (Texas A&M,

USA)

✔ ✔ **A** **009**: Minato, A., Nakajima, N., Nagahara, T., *Simulation of two-phase flow in pumping stations* (Hitachi Corp., Japan)

✔ ✔ **A** **010**: Ninokata, H., *Two-phase flow modeling in rod bundle by subchannel analysis* (TITech, Japan)

✔ ✔ **M** **011**: Lakehal, D., Reboux, S., Liovic, P., *Subgrid-scale modeling for the LES of interfacial gas-liquid flows* (ETHZ Zürich, Switzerland)

✔ ✕ **PI** **012**: Vedrine, D., Donnat, L., *Multiphase flow modelling: how to improve safety & reliability in refineries operation?* (Total-Harfleur, France)

✔ ✔ **N** **013**: Tanguy, S., Ménard T., Berlemont, A., Estivalezes, J. L., Couderc, F., *Développement d'une méthode level set pour le suivi d'interfaces et applications* (Coria Rouen, France, Onera Toulouse, France)

✔ ✔ **N** **014**: Seynhaeve, J.-M., Giot, M., *The WAHA CODE: A numerical tool for water hammer in two-phase flow – Some simulations of experiments* (Université catholique de Louvain, Belgique)

✔ ✔ **N** **015**: Guelfi, A., Hérard, J.-M., Mimouni, S., Ambrsos A., Belliard, M., Emonot, P., Fauchet G., Grandotto, M., Kokh, S., Kumbaro, A., Lemonnier, H., *Key issues for numerical methods in NEPTUNE project* (EDF R&D, and CEA, France)

✔ ✔ **M** **016**: Abbad, M., Caballina, O., Souhar, M., *Etude des effets de mémoire sur des particules sphériquesà faibles nombres de Reynolds* (LEMTA Nancy, France)

✔ ✔ **I** **017**: Leblond, J., J., G., Fournel, B., *Mesure des transferts de masse dans les mousses* (ESPCI, Paris, CEA/Valrho, France)

✔ ✔ **M** **018**: Candelier, F., Souhar, M., *On the effect of the history force on the migration of a bubble and a solid particle in solid body rotation flow* (LEMTA Nancy, France)

✔ ✕ **A(?)** **019**: Sommer de Gélicourt, Y., Grandjean, S., Plasari, E., *CFD modelling of the turbulent precipitation of plutonium oxalate in a vortex reactor* (Politecnico, Torino, Italy, CEA/Marcoule, France, ENSIC, Nancy, France)

✔ ✔ **PI** **020**: Bestion, D., Guelfi, A., *Multiscale analysis of thermal-hydraulics of nuclear reactors - The NEPTUNE project* (CEA/Grenoble, France)

✔ ✕ **I** **021**: Péturaud, P., Hervieu, E., *Physical validation issue of the Neptune two-phase modelling: validation plan to be adopted, experimental programs to be set up and associated instrumentation techniques developed* (EDF R&D, CEA/Grenoble, France)

✔ ✔ **N/M** **022**: Jamet, D., Fouillet, C., Ruyer, P., Klinger, J., *Méthodes à interfaces diffuses pour la modélisation des écoulements multiphasiques* (CEA/Grenoble, France)

✔ ✕ **End** **023**: Jamet, D., Lebaigue, O., Lemonnier, H., *Test-cases for interface tracking methods: Methodology and current status* (CEA/Grenoble, France)

✔ ✔ **N** **024**: Mathieu, B., *Demonstration of a 3D parallel implementation of the front-tracking method: simulation of fluid mixing with a moving boundary and a free surface* (CEA/Grenoble, France)

✔ ✕ **N/M** **025**: Dupuis, A., Yeomans, J., *The dynamics of droplets on chemically and topologically patterned substrates*(Oxford Univ., UK)

✔ ✘ **PI** **[026](#)**: Jacobsen, K., A., Johansen, S., T., *Leda, a next generation tool for prediction of multiphase flows in oil and gas pipelines* (SINTEF Petroleum Reasearch, Norway)

✔ ✔ **N** **[027](#)**: Vincent, S., Caltagirone, J.-P., Delage S., Lacanette, D., Lubin, P., Randrianarivelo, T., *Implicit penalty methods on Eulerian grids for the simulation of incompressible multiphase flows* (TREFLE, Bordeaux Univ., France)

✔ ✔ **A** **[028](#)**: Blais, J.-P., Del Gatto, L., Boudarias, C., Gerbi, S., *Numerical modelling of the mixed flows in hydroelectric schemes* (Electricité de France, CIH, Chambery, France)

✔ ✔ **N** **[029](#)**: Spelt, P., D., M., *Level-set simulations of flows with moving contact lines* (University of Nottingham, United Kingdom)

✔ ✔ **A** **[030](#)**: Raynal, L., *Use of CFD for applications in the oil and gas industry* (Institut Fraçais du Pétrole Solaize, France)

Back to the colloquium home page

177ème session du comité Scientifique et Technique de la
SOCIETE HYDROTECHNIQUE DE FRANCE
25 rue des Favorites - 75015 PARIS - Tél. : 01 42 50 91 03 -
Fax : 01 42 50 59 83 - mail : shf@shf.asso.fr - site www.shf.asso.fr

PROGRES RECENTS DES METHODOLOGIES DE MODELISATION DES ECOULEMENTS DIPHASIQUES
Lyon, France, 24-26 novembre 2004

English version here

Programme définitif

Résumés des communications

Modèle de résumé (MS Word)

Modèle de communication (MS Word)

Inscription

Accès

Visite du LMFA le 24 novembre

La Société Hydrotechnique de France (SHF) et L'Association Internationale pour la Recherche en Hydraulique (AIRH) organisent un séminaire de deux jours sur le thème des progrès récents des méthodologies de modélisation des écoulements diphasiques qui se déroulera à Lyon, France, les 24, 25 et 26 novembre 2004. Les objectifs de ce séminaire sont de rassembler les ingénieurs français et européens autour des nouveaux besoins que requière la mise au point des outils de CFD décrivant les écoulements diphasiques à petite échelle. Certains organismes de recherche et départements de R&D industriels ont lancé des programmes à long terme pour le développement d'une nouvelle génération de modèles et d'outils de CFD. La SHF favorise la mise en contact des « problèmes » et des « solutions » et s'efforce d'animer la communauté francophone de l'hydraulique, notamment en lui offrant des occasions de débattre.

En organisant cette manifestation, la SHF poursuit quatre objectifs.

1. 1. Fournir aux industriels une tribune pour exprimer, devant leurs collègues universitaires ou chercheurs d'autres organismes, les besoins que suscitent leurs nouveaux projets de modélisation des écoulements diphasiques. C'est le cas notamment du CEA, d'Electricité de France et de Total qui sont impliqués dans le développement d'une nouvelle génération d'outils de modélisation tridimensionnels soit dans le cadre du modèle à deux fluides (moyenné en temps) soit à échelle plus fine encore que l'on pourrait rapprocher de celle de la simulation numérique directe.

2. 2. Jean-Marc Delhaye (Université de Clemson, SC, USA) a animé pendant 8 années un groupement de Recherche du CNRS sur les écoulements diphasiques, le GREDIC. Depuis l'échéance de cette structure en 2000, les membres de ce groupe n'ont pas eu l'occasion de se réunir et notre manifestation est une occasion unique de reprendre contact pour tous ses membres.

3. 3. Les préoccupations de l'AIRH et la SHF se recoupent notamment autour des écoulements diphasiques. Toutefois, les deux communautés fréquentent des manifestations différentes et ont peu l'occasion de se retrouver pour discuter ensemble sur ce thème. L'organisation d'une manifestation commune est une occasion d'enrichissement mutuel.

4. 4. Finalement, il est important que l'on puisse identifier les besoins de recherche nécessaires à la réussite des projets industriels. La mise en oeuvre d'une modélisation raisonnable, c'est-à-dire, possédant un sens physique établi, fait appel à des disciplines différentes comme la modélisation physique, le développement d'instrumentation avancée, l'analyse numérique et le développement de techniques expérimentales. La manifestation met l'accent sur les études où l'ensemble de ces développements est réalisé à des échelles cohérentes pour la modélisation tridimensionnelle ou la

simulation numérique directe.

Le français et est la langue officielle de la manifestation. Les contributeurs francophones sont encouragés à proposer leur contribution en français. Les contributeurs non francophones peuvent utiliser l'anglais par souci d'efficacité. Le séminaire comprend deux journées de présentations techniques, le 25 et le 26 novembre, sous forme de sessions constituées de présentations orales de 20 minutes environ suivies de 5 minutes de questions. Le 24 novembre après-midi, une visite du Laboratoire de Mécanique des Fluides et d'Acoustique de Lyon est également au programme. La SHF prépare un recueil des résumés accompagné d'un CD-ROM comprenant les textes des communications qui sera disponible sur le site du séminaire.

Retour en haut de la page

177ème session du comité Scientifique et Technique de la
SOCIETE HYDROTECHNIQUE DE FRANCE
25 rue des Favorites - 75015 PARIS - Tél. : 01 42 50 91 03 -
Fax : 01 42 50 59 83 - mail : shf@shf.asso.fr - site www.shf.asso.fr

ADVANCES IN THE MODELING METHODOLOGIES OF TWO-PHASE FLOWS
Lyons, France, November 24-26, 2004

Version française ici

Final program

Presentation abstracts

Abstract Template (MS word)

Paper Template (MS Word)

Registration

Venue

Visit of LMFA, November 24

The Société Hydrotechnique de France (SHF) and the International Association for Hydraulics Research (IAHR) are organizing a two-day seminar related to the advances on two-phase flow modeling, in Lyons, France on Nov. 25-26, 2004. The objectives of this meeting are to bring together French and also European engineers, scientists and academics to discuss the new needs generated by the use of "small scale" two-phase CFD technology. Several Research Institutes are involved in long-term research programs aimed at developing a new generation of CFD tools and models for two-phase flow and heat transfer modeling. It has always been the continuous role of SHF to put into contact "problems" with "solutions" and to animate the French/European fluid mechanics and Hydraulics community by offering meeting opportunities.

By organizing this meeting, SHF has primarily four main objectives.

1. 1. To provide a tribune to industry to present to our colleagues of academia and other research institutes existing industrial projects on two-phase CFD, at CEA, Electricité de France and Total for example. These projects involve a refined modeling effort towards multiphase 3D time-averaged modeling of two-phase flow and quasi "direct simulation”

2. 2. Since the end of the CNRS-GDR "Gredic" in 2000 (Groupement de recherches sur les écoulements diphasiques), which was exceptionally successful (8 years long) and chaired by Professor Jean-Marc Delhaye of CEA/Ecole Centrale Paris and Clemson University, SC, USA, the French two-phase flow "club" is missing some animation on a regular basis. This meeting may become an opportunity to revive this era.

3. 3. To promote some real technical exchanges between the SHF and AIRH members who usually pertain to slightly disconnected communities but do have common technical issues.

4. 4. To help identifying the research needs in these aforementioned areas in the different disciplines upon which a sound physical model is based and validated, *i.e.* physical modeling, advanced instrumentation development, numerics, and experimental techniques. This scope is of course very wide and the focus is put on studies consistent with the scales relevant to the 3D models or DNS.

French is the official language of the meeting. However non-native French speakers may present their contributions in English for the sake of simplicity. The format of the meeting is two-day with technical sessions consisting in 20-minute oral presentation. The technical sessions will be held on Nov. 25 and 26. A laboratory visit planned on the afternoon preceding the technical sessions, on Nov. 24 (Laboratoire de Mécanique des Fluides et d'Acoustique de Lyon, hosting the seminar). A book of abstracts with a CD-ROM of the full-text manuscripts will be prepared by SHF and will be available at the seminar.

Back to the top of this page

WIRE-MESH SENSORS: AN EXPERIMENTAL TOOL FOR TWO-PHASE CFD MODEL DEVELOPMENT AND CODE VALIDATION

Horst-Michael Prasser[1]
Forschungszentrum Rossendorf e.V., P.O.Box 510119, 01314 Dresden, Germany
prasser@fz-rossendorf.de

Phone: +49 351 260 3460, Fax: +49 351 260 2383, E-Mail: prasser@fz-rossendorf.de

ABSTRACT

The Institute of Safety Research of the Forschungszentrum Rossendorf, Germany, has developed electrode-mesh sensors, which allow the measurement of the electrical conductivity distribution in a flow duct. This can be used either for the detection of the gaseous phase in a gas-liquid flow or for mixing studies in single phase flow, when the components have different electric conductivities. Two grids of crossing wires are placed into the flow closely behind each other. The wires of the first plane (transmitter plane) are supplied with pulses of a driving voltage in a successive order. The data acquisition is done by measuring the electrical currents arriving at the second grid (receiver wires). After the last transmitter electrode has been activated, a two-dimensional matrix is available that reflects the conductivities at crossing points of the electrodes of the two grids. Sequences of these 2D distributions are recorded with a rate of up to 10 kHz.
Due to the high measuring rate each bubble is mapped in several successive instantaneous frames. This allows to obtain bubble size distributions as well as bubble-size resolved gas fraction profiles beside the visualisation and the calculation of profiles of the time-averaged void fraction. Two sensors placed behind each other can furthermore be used for bubble velocity measurements using cross-correlation techniques. Sensors with three layers of electrode grids can be used for the measurement of the velocity of individual bubbles.
The sensor is widely used to study the evolution of the flow pattern in an upwards air-water flow. The experiments aim at closure equations describing forces acting on bubbles as well as coalescence and fragmentation frequencies for the implementation in CFD-codes. The largest sensor used until now has a circular measuring cross-section of about 200 mm diameter and is equipped with two grids of 64 wires. Therefore, the spatial resolution is 3 mm, the measuring frequency is 2.5 kHz. In the meanwhile, a sensor of this kind has been constructed and successfully used in a hot steam-water flow at 70 bar and 286 °C. Experiments were carried out at a vertical test channel of 195 mm inner diameter, in which the distance between gas/steam injection and sensor can be varied in a wide range. Results will be presented.
Some other prominent examples of the application of wire-mesh sensors will be given, like (1) the use of two wire-mesh sensors at the CIRCUS test facility of the University of Delft in the Netherlands for boiling water reactor stability studies, (2) the visualization of cavitation at fast-acting cut-off valves at the Pilot Plant Pipework test facility of Fraunhofer UMSICHT, Oberhausen, (3) the visualization of the flow structure behind a closing globe valve at TU Munich, and finally (4) mixing studies in single-phase flow at the ROCOM test facility in Rossendorf, which are aimed at the mixing of deborated slugs during boron dilution transients. Results will be discussed on basis of animated data visualizations for all examples.
The accuracy and the effect of the wire grids to the flow were investigated using a sensor built into a transparent channel. The comparison with the frames of a high-speed video camera have shown that the sensor acts as a bubble fragmenting obstacle. Nevertheless it was be proved, that the sensor signal represents the bubble geometry present in the upstream flow.

[1] Corresponding author

GAMMA AND X-RAY TOMOGRAPHY FOR TRANSIENT TWO-PHASE FLOWS AND OTHER INSTRUMENTATION DEVELOPED BY ROSSENDORF

Horst-Michael Prasser[1]
Forschungszentrum Rossendorf e.V., P.O.Box 510119, 01314 Dresden, Germany
prasser@fz-rossendorf.de

Phone: +49 351 260 3460, Fax: +49 351 260 2383, E-Mail: prasser@fz-rossendorf.de

ABSTRACT

First subject of the paper is a gamma-tomography setup for imaging a periodically changing density field. It is based on a time-resolved acquisition of the detector signals. The system consists in a 5 Ci source of Cs-137 and a detector arc with 64 BGO scintillation crystals coupled with photo-multipliers. It was used to visualize the gas fraction distribution within the impeller of an axial turbo-pump operating at about 1500 rpm, that delivered a gas-liquid mixture. The detectors operated in pulse mode. The pulses were counted by several banks of 64 counters activated in a successive order - each for a period of 100 µs. The counting procedure is restarted in the first bank after each full rotation of the impeller. After a measuring time of typically 3 -6 min, projections of the density distribution inside the object are acquired in a rotation-angle resolved manner. This allowed to afterwards reconstruct the void fraction distribution inside the rotating impeller structure by applying filtered back-projection algorithms. In a second application, the tomography system was applied to a hydraulic clutch (coupling). These measurements showed the distribution of the hydraulic liquid inside both semi-filled working wheel of the clutch at different slip ratios. An advanced detector system with a total number of 320 crystals is under construction. This will allow to increase the resolution from now about 7 mm to about 2-3 mm inside the measuring plane.

In the field of X-ray tomography, the status of the development of an ultra-fast system based on a scanning electron beam is presented. An electron beam is linearly deflected over a tungsten target with a frequency of 1 kHz. X-rays generated by the traveling focus penetrate the object and arrive at a detector line placed behind the object. The detectors are read-out with a sufficiently high speed in order to obtain projections of the density distribution in different projecting directions, which change thanks to the scanning. First results showing tomographic image sequences of a phantom consisting of small spheres kept in arbitrary motion in a cylindrical test box will be presented. Moving spheres of 3 mm diameter with cylindrical holes of 1 mm diameter were resolved at a framing rate of 1 kHz. For the experiments, an electron beam of a 150 kV gun with a current of 5 mA together with a line of 64 CdZnTe detectors working in current mode with a sampling frequency of 100 kHz for each detector (total data rate: 6.4 MHz) were used. It is planned to continue this development in direction towards an application in two-phase flow experiments.

A second application of X-rays concerns a cone-beam tomography of a stirred vessel reactor. In this case, time-averaged gas fraction distributions produced by a gassing stirrer were visualized using a standard X-ray tube and a 2D detector array. The rotation of the fluid inside the reactor was used to obtain the projections necessary for the 3D reconstruction.

The paper will furthermore present a summary of other kinds of two-phase instrumentation developed in the Research Center Rossendorf. Examples are: (1) local void probes based on impedance measurements, that are equipped with a micro-thermocouple substituting the traditional electrode applied to non-adiabatic gas-liquid flows, (2) the use of high-speed video imaging together with image processing techniques for bubble-column studies, (3) the developments of an optical tomograph and other optical sensors.

[1] Corresponding author

X-RAY IMAGING FOR FLUID FLOW MEASREMENT: PRESENT AND FUTURE TRENDS

Samuel Legoupil
LIST/SSTM, CEA/Saclay, 91191 Gif sur Yvette cedex, France
Phone: +33 (0) 1 69 08 43 13, Fax: +33 (0) 1 69 08 60 30
samuel.legoupil@cea.fr

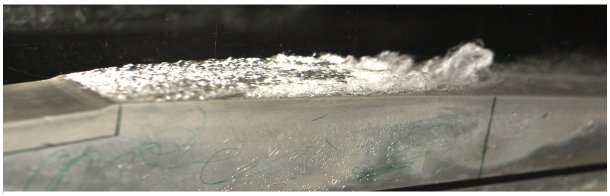
ABSTRACT

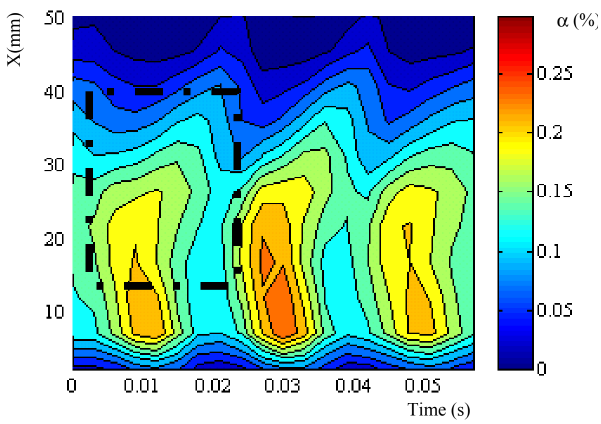
X-rays techniques are widely used in the non-destructive evaluation field for mechanical inspection. However, development of new x-ray detectors and sources over the last decade has let to an intensive use of this technique in other fields. In this paper, we describe the use of X-rays techniques in the field of fluid flow engineering (fluidics and heat transfer). This technique is very attractive since measurements can be performed even if pressure, temperature require the use of opaque walls. In addition, the X-ray technique is well suited to multiphase flows where optical technique can not be used if void fraction is larger than few percents. Specific gravity, mass or void fraction are the main accessible parameters.

The following problem are potential applications for X-ray techniques :

1. Mixing of liquids and gases,
2. Movement of fluids in machinery,
3. Composition of mixtures,
4. Granular flow.

This paper describes the X-ray technique for fluid flow measurement. Depending on the problem, high frequency acquisition time and / or high spatial resolution can be required. Low contrast materials is also a limit to the X-ray technique. Examples will be given in cavitations problem, tomography in mixing process and aircraft lubricant system. This paper will show that high frequencies, up to 200 Hz can be obtained and that high spatial resolution can be obtained.





Void fraction estimation by X-ray attenuation measurement

VOLUME-OF-FLUID METHOD BASED NUMERICAL SIMULATIONS OF GAS-LIQUID TWO-PHASE FLOWS IN CONFINED GEOMETRIES

Martin Wörner, Bradut E. Ghidersa, Milica Ilić, Dan G. Cacuci
Forschungszentrum Karlsruhe, Institut für Reaktorsicherheit,
Postfach 3640, 76021 Karlsruhe, Germany.

ABSTRACT

Within the last decade great progress has been made in the development of advanced numerical methods for computation of gas-liquid flows with deformable interfaces. Among these methods for “direct numerical simulation (DNS)” of incompressible gas-liquid flows, the volume-of-fluid method developed in the eighties has been significantly improved while the level-set and front-tracking methods have newly emerged. The methods mentioned have two main merits. On one hand, they allow to get a deeper insight into the underlying physical mechanisms and thus foster the understanding of gas-liquid two-phase flows. On the other hand they can provide a unique database of the three-dimensional velocity and pressure field and phase distribution with high spatial and temporal resolution which can not be obtained by today’s most advanced experimental techniques. Such a database can be used to develop and improve physical models for CFD codes and engineering flow computations. In this paper we give examples for both merits. Namely, we present computations of the gas-liquid flow within a square mini-channel (see below) and we utilize DNS data of the rise of a bubble swarm in a vertical channel through an otherwise stagnant liquid (Fig. 1) to scrutinize closure assumptions for statistical modeling of bubble-induced pseudo-turbulence.

The direct numerical simulations to be reported in this presentation are based on the volume-of-fluid method for interface tracking, where the phase interface separating the two incompressible fluids is - for each mesh cell that instantaneously contains both phases - locally represented by a plane. The computations are performed by our in-house code TURBIT-VOF, which is based on the finite volume method and employs a staggered, rectangular, structured grid. The governing equations are solved by a projection method and the single field momentum equation is integrated in time by an explicit third-order Runge-Kutta method. Spatial derivates are approximated by second-order central differences.

Due to the potential advantages of micro devices for chemical process engineering two phase flow in narrow channels is of increasing interest. We perform simulations of the co-current bubble train flow of air/oil in a square vertical channel with 2mm width. To account for the influence of the leading/trailing bubble we use in axial direction periodic boundary conditions. By considering two different values of the oil viscosity we investigate the influence of the capillary number, which is the relevant non-dimensional group for gas-liquid flows in narrow channels. The computed values for the bubble diameter and bubble velocity are in good agreement with experimental data from literature. However, our numerical results provide descriptions of the flow at a level of detail that far exceeds the insight one can get from experiments alone (Fig. 2). We find that a decrease of the capillary number by a factor of five is associated with the thinning of the liquid film, the appearance of a second vortex in the bubble, the enhancement of non-axisymmetry of the flow inside the bubble, and the appearance of a vortex in the liquid slug.

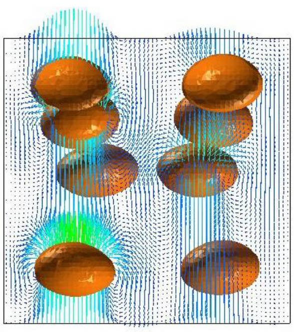


Fig. 1: Snapshot of instantaneous bubble shapes and velocity field in a wall-normal plane for the bubble swarm simulation.

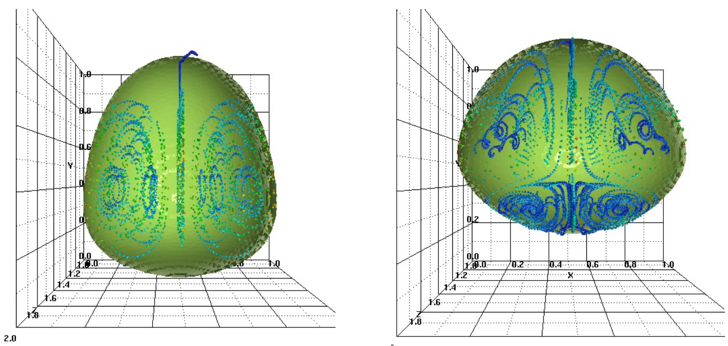


Fig. 2: Visualization of steady bubble shape and flow structure inside the bubble rising in a square vertical mini-channel for two different values of the capillary number.

DYNAMIC MODELING OF INTERFACIAL STRUCTURES VIA INTERFACIAL AREA TRANSPORT EQUATION

Seungjin Kim
Nuclear Engineering Department
1870 Miner Circle, University of Missouri – Rolla
Rolla, MO 65409-0170, USA

Mamoru Ishii
School of Nuclear Engineering
400 Central Drive, Purdue University
West Lafayette, IN 47907-1290, USA

ABSTRACT

In the current thermal-hydraulic system analysis codes using the two-fluid model, the empirical correlations that are based on the two-phase flow regimes and regime transition criteria are being employed as closure relations for the interfacial transfer terms. Due to its inherent shortcomings, however, such static correlations are inaccurate and present serious problems in the numerical analysis. In view of this, a new dynamic approach employing the interfacial area transport equation has been studied. The interfacial area transport equation dynamically models the two-phase flow regime transitions and predicts continuous change of the interfacial area concentration along the flow field. Hence, when employed in the therma-hydraulic system analysis codes, it eliminates artificial bifurcations stemming from the use of the static flow regime transition criteria. Therefore, the interfacial area transport equation can make a leapfrog improvement in the current capability of the two-fluid model from both scientific and practical point of view.

Accounting for the substantial differences in the transport phenomena of various sizes of bubbles, the two-group interfacial area transport equations have been developed. The group 1 equation describes the transport of small-dispersed bubbles that are either distorted or spherical in shapes, and the group 2 equation describes the transport of large cap, slug or churn-turbulent bubbles. The source and sink terms in the righthand-side of the transport equations have been established by mechanistically modeling the creation and destruction of bubbles due to major bubble interaction mechanisms. The coalescence mechanisms include the random collision driven by turbulence, and the entrainment of trailing bubbles in the wake region of the preceding bubble. The disintegration mechanisms include the break-up by turbulence impact, shearing-off at the rim of large cap bubbles and the break-up of large cap bubbles due to surface instability.

In the present paper, the interfacial area transport equations currently available are reviewed to address the feasibility and reliability of the model. Results from extensive benchmark experiments for the model evaluation are also present. These include the data from adiabatic upward air-water two-phase flow in round tubes of various sizes, from a rectangular duct, and from adiabatic co-current downward air-water two-phase flow in round pipes of two different sizes. Furthermore, some guidelines for the future study on interfacial area transport equation are discussed.

SUPERCRITICAL SEWER FLOW – RECENT ADVANCES

Willi H. Hager
VAW, ETH-Zentrum, CH-8092 Zurich, Switzerland[1]
Phone: +41 1 632 41 49, Fax: +41 1 632 11 92, e-mail: hager@vaw.baug.ethz.ch

Corrado Gisonni
Università di Napoli 2, Via Roma 21, I-81031 Aversa, Italy
gisonni@unina.it

ABSTRACT

Supercritical flow in open channels is characterized by the generation of shockwaves due to any deviation from the prismatic channel alignment. These standing waves determine the required freeboard and constrain the discharge capacity, and thus are an important design characteristic. Combined sewers must hydraulically correspond to partially filled pipe flow to reduce problems with air-water two-phase flow features that are beyond a basic hydraulic design procedure. These flows may be described as stratified composed of a lower water, and an upper co-current air flow. Current design criteria require a maximum filling ratio of 85%, depending on national standardization.

The paper to be presented summarizes recent experimental findings on supercritical sewer flow. Of particular interest are the flow patterns across sewer manholes because of the formation of shockwaves. If a sewer does not correctly account for high-speed flow, then transition from stratified to choking flow occurs, corresponding to an abrupt transition from free surface to pressurized air-water flow. The paper describes the main features of flows both in manholes and in straight sewers by including the bend and the junction manholes. It will also be demonstrated that small drops in a manhole are a poor design. The oral presentation describes typical flow scenarios using figures and plots, whereas the paper intends to highlight the current design approach, based on extended experimental observations and computational analysis.

[1] Corresponding author

THE DIRECT NUMERICAL SIMULATION OF TWO-PHASE FLOWS WITH SHARP INTERFACE CAPTURING METHODS

Sanjoy Banerjee
Department of Chemical Engineering, Department of Mechanical Engineering

University of California, Santa Barbara, CA 93106
banerjee@engineering.ucsb.edu

J.C. Nave
Department of Mechanical Engineering

University of California, Santa Barbara, CA 93105
jcnave2@hotmail.com

Sanjoy Banerjee, Department of Chemical Engineering, University of California, Santa Barbara, Santa Barbara, CA 93106 Phone: 01-805-893-3456, Fax: +01 805-893-4731, E-Mail: banerjee@engineering.ucsb.edu

ABSTRACT

Multiphase flows play a central role in problems related to the environment and industry. Common, but significant, features of such problems are complex geometries and topography, transport processes across phase boundaries, and internal interfaces that merge, break, and deform. Requirements to predict fluid motion under such conditions provide significant challenges for computational fluid dynamics. In this paper, we will focus on two models that capture the motion of internal interfaces implicitly and can resolve complex flows down to computational grid sizes. The methods are based on the phase-field approach, e.g., Model H of Hohenberg and Halperin. For processes occurring over very small length scales, e.g. nucleation and growth, such computations result in finite thickness interfaces and capture the necessary physics. For larger length-scale multiphase structures, the interface becomes a contact discontinuity, and the method smoothly transitions into a level-set-like formulation. However, in this second form the sharp interface is computationally difficult to handle. A variation of the ghost-fluid method, in which sharp interfaces can be captured, will be presented. Applications will be shown at the two extremes of scales, including early stages of nucleation, growth and coarsening of multiphase structures, as well as macroscopic flows in which the length-scales and velocities are large enough to produce turbulence. In the second case, the interaction of turbulent structures with interfaces will be captured in direct numerical simulations. As the interfaces are sharp, computations of heat and mass transfer are possible without being rendered inaccurate due to regions where the interface is smeared over several grid points. Also, a general strategy for computation of multiphase flows will be discussed in which supergrid structures are resolved and subgrid structures modeled. Comparisons with experiments will be discussed to validate the procedures presented.

DRAG REDUCTION VIA MICROBUBBLE INJECTION IN BOUNDARY LAYER OF CHANNEL FLOWS

Yassin A. Hassan
Department of Nuclear Engineering
Texas A&M University

College Station, Texas 77843-3133 USA
Phone: +01 979 845 7090, Fax: +01 979 845 6443, E-Mail: y-hassan@tamu.edu

ABSTRACT

Methods to reduce the drag in turbulent flows have been carried out for the past several decades. Reducing skin friction has obvious advantages through improvements in fuel economy, range (as in case of commercial ships and aircraft) or in peak speed (desirable in military or racing applications) and for less impact on the environment due to less fuel consumption. Recently, the reduction of turbulent friction between a solid surface and fluid by adding drag reducing additives has received increasing attention for saving power and reducing the pollution. Polymers and surfactants injections, wall oscillations, traveling waves, blowing and suction, and microbubble injection are examples among active additives. Riblets are an example of passive techniques to achieve drag reduction. However, a consensus about understanding the mechanism that governs this phenomenon has not been reached. In this paper, an investigation of turbulent structure modification of fully developed channel flow by microbubble injection close to the upper wall was studied. Two-dimensional velocity components at Reynolds number of 5128 based on the half height of the channel and bulk velocity were measured. The particle image velocimetry technique was utilized to obtain the two-dimensional velocity fields of the fluid and the microbubbles. Microbubbles with an average diameter of 30 um were injected into the buffer layer. Various values of void fractions were used to evaluate the effects of microbubbles concentration on the drag reduction. Modifications in the length and time scales were detected by calculating two-point correlation coefficients. Streamline length and time scales were increased. On the contrary, the normal length and time scales were decreased with the increase of the drag reduction. The presence of the microbubbles with low local concentration of 4% achieved 40% drag reduction. A decrease in the Reynolds stresses were achieved as the void fraction increased. The Q_4 events (sweeps) which are responsible for the production of turbulence were reduced. The energy spectra were calculated. It is interesting to note a shift from larger wave number to lower wave number near the wall region.

SIMULATION OF TWO-PHASE FLOW IN PUMPING STATIONS

Akihiko Minato[1], Nobuyuki Nakajima
Hitachi Ltd., 7-2-1 Omika, Hitachi, Ibaraki 319-1221, Japan
Phone: +81 (294) 55-8074, Fax: +81 (294) 53-7664, E-Mail: akihiko_minato@pis.hitachi.co.jp, nobuyuki_nakajima@pis.hitachi.co.jp

Takahide Nagahara
Hitachi Industries Co., Ltd., 603 Kandatsu, Tsuchiura, Ibaraki 300-0013, Japan
Phone: +81 (29) 832-8128, Fax: +81 (29) 832-8002, E-Mail: takahide_nagahara@gm.hitachi-hic.jp

ABSTRACT

The purpose of the present study is to develop numerical tools for two-phase flow problems in pumping stations. Reliable simulation methods of two-phase flow are inevitable in nuclear engineering field because computer programs are important design tools for nuclear reactors which are exposed to severe ambient conditions of high temperature, high pressure and gamma ray irradiation. Mock-up tests of nuclear thermal-hydraulics are very difficult. Hitachi Ltd. has created advanced computer programs for multi-dimensional two-phase flow to analyze reactor systems and components. The numerical methods enable accurate and stable interface calculation in spite of large density ratio between the phases, because of non-staggered finite volume method, which is suitable for treatment of discontinuity, and minimizing numerical diffusion. The multi-dimensional computer programs employ two types of two-phase flow model. The first one is the extended two-fluid model, which has capability of both interface tracking and ordinary two-fluid model calculations. This model is appropriate for the present purpose because it can treat free surface of open channels and entrained bubbly mixture due to breaking waves. The second one is the improved VOF (Volume of Fluid) method in which PLIC (Piecewise Linear Interface Calculation) technique was employed. Local interface state calculation is simplified by approximating cubic calculation cells to spherical ones. These computer programs were applied to hydraulic studies in Hitachi Industries Co., Ltd., a manufacturer of fluid machinery and pumping systems. It is necessary to evacuate air downstream from siphon outlets as bubble entrainment by water flow in order to operate the plant effectively after pumping station start-up. Experiments were carried out by using a reduced scale test-section. The test section was made of transparent acryl and the cross-section was rectangular. Free surface of water layer on the inclined wall and entrained bubbly mixture into the water pool due to water layer collision with free surface were observed by a video camera. The extended two-fluid model results of transient water level and pressures in the simulated siphon outlets were in good agreement with the experimental data. The predicted pressure histories at the crest and the pump site were in good agreement with the experimental data. The experiments of sedimentation in an inlet channel of a pump station was carried out. Three dimensional free surface flow was calculated by the improved VOF method and behavior of suspended sediment in the flow field was calculated. The surface wave velocity in the shallow open channel was in good agreement with the theoretical one. Sediment particles had size distribution, and drag force, gravity and buoyancy were taken into consideration. Critical tractive force of the sediment on floor was estimated from Schields' correlation. The predictions of sedimentation area and suspended fraction agreed well with experimental data. These results showed that the present computer programs were able to treat key two-phase flow phenomena in pumping stations.

[1] Corresponding author

TWO-PHASE FLOW MODELING IN THE ROD BUNDLE SUBCHANNEL ANALYSIS

Hisashi Ninokata
Tokyo Institute of Technology
2-12-1 O-okayama, Meguro-Ku, Tokyo 152-8550 Japan
hninokat@nr.titech.ac.jp
Phone/FAX: +81 (3) 5734-3056

ABSTRACT

In order to practice a design-by-analysis of thermohydraulics design of BWR fuel rod bundles, the subchannel analysis would play a major role. There, the immediate concern is improvement in its predictive capability of CHF due in particular to the film dryout (boiling transition phenomena: BT) on the fuel rod surface. Constitutive equations in the subchannel analysis formulation are responsible for the quality of calculated results. The constitutive equations are a result of integration of the local and instantaneous description of two-phase flows over the subchannel control volume. In general, they are expressed in terms of subchannel-control-volume- as well as area-averaged two-phase flow state variables. In principle the information on local and instantaneous physical phenomena taking place inside subchannels must be counted for in the algebraic form of the equations on the basis of a more mechanistic modeling approach. They should include also influences of the multi-dimensional subchannel geometry and fluid material properties. Thermohydraulics phenomena of interests in this deed are: 1) vapor-liquid re-distribution by inter-subchannel exchanges due to the diversion cross flow, turbulent mixing and void drift, 2) liquid film behaviors, 3) transition of two-phase flow regimes, 4) droplet entrainment and deposition and 5) spacer-droplet interactions. These are considered to be five key factors in understanding the BT in BWR fuel rod bundles.

In Japan, a university-industry consortium has been formed under the sponsorship of the Ministry of Economics, Trade and Industry. This paper describes an outline of the on-going project and, first, an outline of the current efforts is presented in developing a new two-fluid three field subchannel code NASCA being aimed at predicting onset of BT, and post BT phenomena in advanced BWR fuel rod bundles including those of the tight lattice configuration for a higher conversion. Then the current methodology adopted to improve the NASCA code capabilities for BT is described. There a combination of experimental and computational fluid dynamics approaches is undertaken to construct a two-phase fluid dynamics database. The experimental approach consists of 1) high-resolution air-water tests performed under the room-temperature and atmospheric pressure conditions for the inter-subchannel exchanges, three-dimensional behaviors of liquid films, and spacer effects; and 2) integral steam-water tests performed at high-temperature and at higher pressure. In the integral tests, state-of-the-arts of multi-phase flow measurement technologies are applied in order to obtain local and instantaneous data that reveal underlying detailed physical processes including high resolution void distributions inside a 4x4 bundle, liquid film thickness and two-phase flwo regime.

The analytical approach consists of computational multi-phase fluid dynamics (CMFD) applicable to two-phase flows. A physical interpretation of the equilibrium two-phase flow redistribution inside a rod bundle is discussed that is considered to closely be related to the void drift phenomena. Identification of interactions among dominant factors is a main objective of the integral test and acquired data will be utilized in verifying the improved subchannel code. Construction of a complete set of two-phase fluid dynamics database will be made by supplementing missing data regions with the aid of numerical analyses. Dependency on important state variables is extracted from the database and prototype constitutive equations are going to be proposed in the final stage of the project.

MULTI-SCALE TURBULENCE MODELLING FOR THE LES OF INTERFACIAL GAS-LIQUID FLOWS

Djamel Lakehal
Institute of Energy Technology, ETH Zurich

ETH Zentrum CLT, CH-8092, Switzerland
Phone/Fax: +49 (0)1 6327073, E-Mail: lakehal@iet.mavt.ethz.ch

ABSTRACT

Traditionally, turbulent multi-fluid flows have been simulated as inter-penetrating media, using the averaged two-fluid representation for void fraction distribution and the Reynolds-averaged Navier-Stokes (RANS) framework for turbulence modeling. In terms of interface topology and turbulence modeling, the two-fluid and RANS approximation approaches are equivalent, in that interface and turbulence structures are not directly generated as part of the solution, but rather modeled. Phase averaging in the two-fluid formalism is not ideal for capturing interface-turbulence interactions, because interface jump conditions cannot be tractable if the phases are mixed. In the case of interfacial multi-fluid flows, the interface location can be tracked only by means of the single-fluid method. Modern *interface-tracking* techniques are indeed capable of providing accurate solutions for interface dynamics within finite-volume/difference simulations, with the premise of tackling surface-turbulence interactions in flows involving abrupt topology changes.

In this paper we introduce the theoretical basis for the LES approach for turbulent multi-fluid flows in general, followed by a critical discussion of its usefulness and limitations. The idea behind it relies on the density-based filtered single-fluid Navier-Stokes equations. The super-grid interface kinematics and turbulence are fully resolved, whereas the unresolved sub-grid scale (SGS) turbulence and interfacial scales are modelled. The method has been recently applied to capture interface-turbulence interactions in the downward gas injection into a water pool, and wave breaking generated turbulence in spilling breakers. In both applications use was made of the Smagorinsky model for unresolved turbulence scales, which proved highly dissipative, affecting near-interface regions in particular. A first attempt to regularize the model by introducing a near-interface damping function turned out to be highly CPU demanding. The dynamic approach of Germano *et al*. [*Physics of Fluids*, **A3**, 1991] cannot be extended to non-homogeneous flows, where there is no specific direction for averaging.

In the present contribution we report on the recent developments achieved by exploring the Variational Multi-Scale Model (VMS) of Hughes *et al*. [*Physics of Fluids*, **13**(2), 2001]. The methodology has been applied to the stratified air-water flow studied by Fulgosi *et al*. [*JFM*, **482**, 2003], for which extensive DNS data are available for comparison. The VMS idea, which relies on projection rather than smoothing, consists in decomposing the energy spectra into three portions: large, small and unresolved. The large and small parts are solved directly whereas the unresolved part is approximated, using, for example, the classical Smagorinsky model. The small-scale portion of the spectra perceives the effect of the unresolved scales only. The model has been incorporated into the same pseudo-spectral solver used by Fulgosi *et al*., in which the interface deformations are simulated by means of the Boundary-Fitting method. The remarkable results obtained will be presented and discussed.

MULTIPHASE FLOW MODELLING: HOW TO IMPROVE SAFETY & RELIABILITY IN REFINERIES OPERATION?

D. Védrine, L. Donnat
TOTAL France
CReG/Process & Refining Division
Po.Box 27, Harfleur 76700, FRANCE
denis.vedrine@total.com, ludovic.donnat@total.com
Phone: +33 (0)2 35 55 13 60, Fax: +33 (0)2 35 55 14 70

ABSTRACT

Since the beginning of the 1990's TOTAL has used CFD software packages to understand the main hydrodynamic phenomena inside the units, by an analysis of the flow patterns and, thereby, to identify the cause of a problem and evaluate technological solutions. More than 150 studies have been achieved during the last 12 years and industrial applications are now frequent, especially since the last 5-6 years, thanks to model developments, calculation time reducing and first industrial feedbacks. Some of the equipments of the major refining processes (Fluid Catalytic Cracking, Alkylation, Hydrotreating, Reforming, … etc.) have now been studied using CFD, allowing to push back units constraints and to improve safety & reliability. Today, the main challenge is to describe more accurately complex multiphase phenomena (coalescence&break-up, emulsions, flow regimes transition, ...) and, in a second step, to closely couple them with the specific physics of each process (essentially chemical reaction, heat and mass transfer).

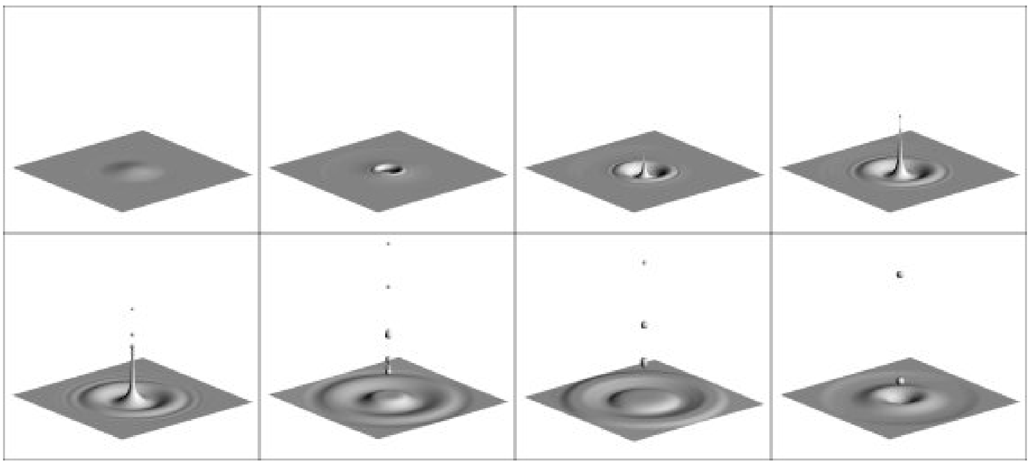
LEVEL SET METHOD FOR INTERFACE TRACKING: DEVELOPMENT AND APPLICATIONS

Sébastien Tanguy,Thibaut Ménard, Alain Berlemont[1],
firstname.name@coria.fr
Phone : (33) 2 32 95 36 17

Jean Luc Estivalezes, Frédéric Couderc
firstname.name@onecert.fr
Phone: (33) 5 62 32 25 28 32

ABSTRACT

Numerical simulation for multiphase flows requires a specific approach to describe interface behaviors, such droplet coalescence or break-up. Front tracking methods are based on the Lagrangian tracking of marker particles; they are more efficient when the interface curvature does not exhibit stiff behaviors. However it appears that topological changes, which are involved in droplet coalescence, depend on an interaction time parameter of great influence. Volume of Fluid method is describing the volumetric fraction of each phase in grid cells. The main disadvantage of the method is the interface reconstruction that appears quite difficult on 2D domain, and numerically prohibitive on 3D domain. A consequence is the uncertainty on interface curvature and thus on surface tension forces. Our work deals with a Level Set method that describes the interface with the zero level curve of a continuous function. This function is defined as the signed distance to the interface. Its advancement is updated with a convection equation, coupled with a constraint that ensures that the function is always a signed distance. Different numerical methods can be implemented to take into account for surface tension forces (delta formulation or ghost fluid method), which depend on the curvature of zero level curve, in Navier Stokes equations. We provide the advantages and the drawbacks of this method and we discuss about its potential improvements. The main target of this study is to reach a better understanding of primary and secondary atomization processes, studying jet stability and droplets collisions.



Level set method application: Free surface bubble burst

[1] Corresponding author

THE WAHA CODE : A NUMERICAL TOOL FOR WATER HAMMER IN TWO-PHASE FLOW – SOME SIMULATIONS OF EXPERIMENTS

Michel GIOT & Jean-Marie SEYNHAEVE
giot@term.ucl.ac.be, seynhaeve@term.ucl.ac.be
Phone: +32 (0) 10 472 233, Fax: +32 (0) 10 452 692, E-Mail: dupont@term.ucl.ac.be

ABSTRACT

The WAHA computer code has been developed during the WAHALoads project in the frame of the 5th European Program. The main purpose of the WAHA code is to predict various mechanisms of single- and two-phase water hammer transients in piping systems. The WAHA code can simulate thermal-hydraulic transients with one-dimensional six-equation two-fluid model approximations and calculate forces on the piping the hydraulic forces on the piping system due to water hammers in two-phase flow.

In the first part of the present paper, the main features of the WAHA code and its capabilities are presented: basic equations, specific closures laws, numeric scheme, possible boundary conditions, some basic results of water hammer induced by a rapid valve closure.

In the second part of the paper, a description of the UMSICHT test facility **PPP** which has been used for water hammer tests inducing bubble condensation is given. The installation has been equipped with a new measurement technology including new wire mesh sensors with thermocouples for the measurement of condensation heat transfer and other innovative transient measurement techniques. From the UMSICHT data bank, some basic data have been chosen as reference case for validating the WAHA code.

The third part dealts with the simulation of several test cases using the WAHA code :

- Edwards pipe - Discharge of a hot liquid from a horizontal pipe.
- **PPP** water hammer tests – Depressurisation inducing cavitation water hammer initiated by a fast valve closure.
- Transient flow induced by a full break of a piping sytem.

The results of these simulation are systematically compared with the experimental data, when available. It is concluded that the WAHA code has the abilitity to correctly predict various mechanisms of transients flows occurring in piping system.

KEY ISSUES FOR NUMERICAL METHODS IN NEPTUNE PROJECT

Antoine Guelfi[1,#], Jean-Marc Hérard[1,*] , Stephane Mimouni[1]
[1] EDF-DRD / MFTT, 6 quai Watier 78400 Chatou, France
Antoine.Guelfi@edf.fr, Jean-Marc.Herard@edf.fr , Stephane.Mimouni@edf.fr

Annalisa Ambroso[2], Michel Belliard[3], Philippe Emonot[4], Gauthier Fauchet[4],
Marc Grandotto[3], Samuel Kokh[2], Anela Kumbaro[2], Hervé Lemonnier[4]
[2] CEA, Centre de Saclay, 91191 Gif sur Yvette Cedex, France,
[3] CEA, Centre de Cadarache, 13108 Saint Paul lez Durance cedex, France,
[4] CEA, Centre de Grenoble, 17 rue des Martyrs, 38054 Grenoble cedex 9, France

firstname.name@cea.fr

ABSTRACT

We will try to present herein the main issues of our investigation in numerical methods for two-phase flow modeling, within the framework of the **NEPTUNE** project, which benefits from both contributions of CEA and EDF. These may be recast in five work packages. The first two are devoted to the mathematical and numerical modeling of two-phase flows with interfaces and the two-fluid two-pressure approach. This in particular includes investigation of relaxation methods in order to establish correct links with standard two-fluid models, which are the core of the third work package. Computations of the interaction of shock waves with bubbles will be presented. Some new results concerning two-fluid and three-field flow modeling will also be briefly presented. Part of the work in the third work package concerns benchmarking, and comparison with several hyperbolic solvers, but also improvement of numerical treatment of source terms, multi-field models and suitable boundary conditions. The fourth one, which deals with the interfacial coupling of codes, is probably the most important one since it requires connecting all models together. Since little attention has been paid to this crucial point, part of the focus will be given in this paper on the coupling of equations of state, one-dimensional and three-dimensional codes, porous approach and free medium approach, but also on ongoing work concerning relaxed and unrelaxed hyperbolic two-phase flow models. The fifth work package gathers all classical contributions in numerical methods, including: recent applications of fictitious domain methods ; preconditioning of so-called "low Mach number" two-phase flows (with applications to the motion of rising bubbles in water) ; parallel and multigrid techniques (with applications to steam generators in nuclear power plants) ; Finite Volume Element methods (with applications to the standard two-fluid models) ; construction and validation of new exact or approximate Riemann solvers (in order to cope with vanishing phases). The latter five obviously aim at improving accuracy, stability and also at reducing CPU time. A few examples will enable to highlight the main advantages and possible drawbacks of these new developments, and the impact of the current and future increasing computational facilities.

Main past achievements, and key points of current and future work on all these issues will be discussed. All available references will be given in order to help the reader getting a more accurate insight on these various contributions. The whole has clearly benefited from contributions of several PhD students : Thomas Fortin, Vincent Guillemaud, Olivier Hurisse, Angelo Muronne, Isabelle Ramière, Jean-Michel Rovarch and Nicolas Seguin.

[#] NEPTUNE project manager
[*] corresponding author (also in CNRS, LATP, UMR 6632, 39 rue Joliot Curie, 13453 Marseille cedex 13. Email: herard@cmi.univ-mrs.fr)

MEMORY EFFECTS ON SPHERICAL PARTICLES AT LOW AND INTERMEDIATE REYNOLDS NUMBERS

Mustapha Abbad
LEMTA, UMR7563 (CNRS-INPL-UHP)
ESSTIN 2, rue Jean Lamour, 54519 Vandoeuvre Cedex, France
mustapha.abbad@esstin.uhp-nancy.fr

Ophélie Caballina[1], Mohamed Souhar
LEMTA, UMR7563 (CNRS-INPL-UHP)
ENSEM 2, avenue de la Forêt de Haye, BP 160, 54504 Vandoeuvre Cedex, France
ophelie.caballina@ensem.inpl-nancy.fr, mohamed.souhar@ensem.inpl-nancy.fr
[1]Phone: +33 (0)3 83 59 56 69, Fax: +33 (0)3 83 59 55 51

ABSTRACT

The present work deals with the recent advancements achieved in the modeling of two-phase flows with dispersed phase. During the two last decades, such flows are of growing occurrence in many industrial applications, such as spray combustion, atmospheric dispersion pollutants, and chemical engineering processing. In the frame of a lagrangian approach, the efficiency of numerical predictions of such phenomena lies in the accuracy of modeling the behavior of the dispersed phase, which requires, mainly, the knowledge of all hydrodynamic forces acting on one particle. Among them, the buoyancy force, the quasi-steady drag, the added mass force and the lift force are currently included and their adequate expressions are now well defined. Nevertheless, the history force, taking into account the vorticity diffusion in the surrounding fluid and the disturbance effect caused by the acceleration of the sphere, is often neglected in the momentum balance.
The results presented herein come from an original experimental study on the effect of the history force acting on a fluid or solid inclusion having an accelerated motion in a viscous incompressible Newtonian medium. In an oscillating frame, the instantaneous position of the free-falling sphere is recorded using a high-speed video camera. The trajectory is then extracted using techniques of image processing based on contour detection. The average terminal velocity, the oscillation magnitude and the phase shift with the vibrating plate are then provided. The trajectories are obtained for Reynolds numbers less than 10 and finite Stouhal numbers ranging from 1 to 20. A comparison is made by solving the equation of motion of the sphere with and without the history integral. In a first step, the experimental data emphasize the relevant role played by this force which gives a more precise prediction of the particle velocity and thus its trajectory. In a second step, several formulations of the kernel used in the integral expression of the history force are examined, especially that of Basset (1888) for creeping flow limit, and that of Mei (1994) for finite Reynolds numbers. The recent results obtained for rigid particles in the mentioned range of Reynolds number and Strouhal number show that the history force is well predicted by the Basset expression.

[1] Corresponding author

MASS TRANSFER MEASUREMENTS IN FOAMS

Jacques, J.G., Leblond[1]
ESPCI, Lab. de Physique Thermique, 10 rue Vauquelin, 75231 Paris cedex 05, France
jacques.leblond@espci.fr

Bruno Fournel
DTCD/SPDE/LFSM, CEA VALRHO, 26702 Pierrelatte cedex, France
bruno.fournel@cea.fr

ABSTRACT

This study participates to the elaboration of a method for decontamination of the inside surfaces of steel structures (pipes, tanks,...).
The solution which has been chosen is to attack the surface of the structure by a dipping solution.
In order to reduce the quantity of product to be recovered and treated at the end of the cleaning process, the active solution will be introduced as a foam. During its free or forced drainage the foam supplies an active liquid film along the structure surfaces.
It was important to know if the transfers of the dipping liquid inside the foam and between foam and wall film are sufficient to allow a correct supplying of the active liquid at the wall and a correct dragging of the dipped products.
The objective of this work is to develop a numerical model which simulates the various transfers.
However such a modeling cannot be performed without a thorough knowledge of the different transfer parameters in the foam and in the film.
The following study has been performed on a model foam (foaming water + air) held in a smooth vertical glass pipe and submitted to a forced drainage by the foaming water (water + surfactants).
.
The liquid transfer involves the dispersion of the drainage liquid inside the foam and the transfer between the foam and the liquid film flowing down at the wall.
The different transfers has been analyzed by IRM using a PFGSE-NMR sequence, which allows to determine the propagator, i.e., the probability density of the liquid particle displacements during a given time interval Δt, along a selected direction.
This study allowed to measure, firstly, the mean liquid and the liquid dispersion in the foam along the vertical and horizontal direction, and secondly, the vertical mean velocity in the parietal liquid film.

[1] Corresponding author

EFFET DE LA FORCE D'HISTOIRE SUR LA MIGRATION D'UNE PARTICULE DANS UN ECOULEMENTS EN ROTATION SOLIDE, A FAIBLE NOMBRE DE REYNOLDS.

Fabien Candelier[1], Mohamed Souhar
LEMTA, UMR7563 (CNRS-INPL-UHP)
ENSEM 2, avenue de la Forêt de Haye, BP 160, 54504 Vandoeuvre Cedex, France
fabien.candelier@ensem.inpl-nancy.fr et mohamed.souhar@ensem.inpl-nancy.fr
[1]Phone: +33 (0)3 83 59 57 12, Fax: +33 (0)3 83 59 55 51

RESUME

Il est assez bien connu qu'une particule plongée dans un écoulement en rotation solide subit une migration radiale dirigée vers le centre de la rotation, si la particule est plus légère que le fluide environnant, ou vers l'extérieur dans le cas contraire. Ce phénomène physique est d'ailleurs largement exploité dans l'industrie, au travers, par exemple, des centrifugeuses ou encore, des séparateurs de phases. Pourtant, malgré l'apparente simplicité de cet écoulement, plusieurs questions reste ouvertes, comme par exemple, la nature et l'influence respective des différentes forces qui s'exercent sur la particule. En effet, qu'il s'agisse d'une bulle, d'une goutte ou d'une sphère solide, le mouvement de la particule est instationnaire et de ce fait, elle subit une force dîte d'histoire.

Dans cette étude, nous nous intéresons principalement à l'effet de cette force sur la migration de la particule. Pour cela, un dispositif expérimental permettant de générer un écoulement en rotation en solide et d'obtenir de manière précise des trajectoires de particules a été réalisé au Lemta. Les résultats expérimentaux obtenus sont confrontés à la solution analytique de l'équation de mouvement d'une particule solide, puis d'une bulle et d'une goutte, prenant en compte la force d'histoire. Les résultats théoriques révèlent une forte influence de la force d'histoire et sont en très bonne adéquation avec les résultats expérimentaux.

[1] Corresponding author

CFD MODELING OF THE TURBULENT PRECIPITATION OF PLUTONIUM OXALATE IN A VORTEX REACTOR

Yann SOMMER de GELICOURT[1]
Dip. Scienza dei Materiali e Ingegneria Chimica
Politecnico di Torino, Corso Duca degli Abruzzi, 24 - 10129 Torino
Phone : +39.011.5644679, Fax : +39.011.5644699
yann.sommerdegelicourt@polito.it

Stéphane GRANDJEAN
CEA VALRHÔ, DRCP/SCPS/Laboratoire de Chimie des Actinides
Bât.399, BP 10171 – 30207 Bagnols sur Cèze
Phone : +33(0)4.66.79.16.03, Fax : +33(0)4.66.79.65.67, E-Mail : stephane.grandjean@cea.fr

Edouard PLASARI
ENSIC, Laboratoire des Sciences du Génie Chimique
1, rue Grandville – 54001 Nancy
Phone: +33(0)3.83.17.50.99, Fax: +33(0)3.83.17.50.86, E-Mail: edouard.plasari@ensic.inpl-nancy.fr

ABSTRACT

The nuclear fuel reprocessing, as it is done in La Hague COGEMA's plant, involves a precipitation in an unbaffled stirred tank to turn plutonium nitrate into plutonium oxalate prior to a calcination for using it in MOX fuel (Mixed Oxydes). In every crystallization and precipitation (reactive crystallization), the key variable is *sursaturation*, resulting from either a chemical reaction between two liquids, a liquid and a solid, a gas and a solid, or a decrease in the product solubility induced by temperature gradient or mixing of a solvent and an anti-solvent. According to the *local sursaturation degree*, different mechanisms may occur leading to the solid dispersed phase : nucleation, growth and eventually aggregation, breakage and ripening. The challenge in that kind of modeling lies in the non linear behavior of the physical and chemical phenomenons, and in the different time and length scales involved which can not be solved without resorting to computational fluid dynamics (CFD). The modeling philosophy used here is divided in two parts : (1) a lower mesh statistical mixing model (FM-PDF) and a liquid energy spectrum model coupled to the RANS equations, for taking into account the continuous liquid phase mixing at various levels : reactor scale, turbulent dispersion and molecular diffusion ; (2) a population balance model for the solid dispersed phase to link flow pattern to the morphology and chemical properties of particles. Many approaches have been developed so far to solve PBE but the recent breakthrough that allowed implementation of that equation in CFD codes is QMOM : quadrature method of moments, that represents the particle size distribution (PSD) in a finite number of delta functions, corresponding to abscissas and weights of the quadrature, which are used to calculate the mean particle size or whatever moment of the PSD. Part (1) has been successfully validated experimentally by means of an acid-base neutralization reaction ; part (2), i.e. the PBE modeling coupled to experimental nucleation and growth laws and an aggregation kernel based on local shear rate, has given satisfactory results, strengthening the promising abilities of this local approach.

[1] Corresponding author

MULTISCALE ANALYSIS OF THERMAL-HYDRAULICS OF NUCLEAR REACTORS - THE NEPTUNE PROJECT

Dominique Bestion
CEA-GRENOBLE, DEN/DER/SSTH, 17 rue des martyrs, 38054 GRENOBLE CEDEX 9
Phone: +33 (0) 4 38 78 36 45, Fax: +33 (0) 4 38 78 51 95, E-Mail: dominique.bestion@cea.fr

ABSTRACT

The NEPTUNE project aims at building a new two-phase thermalhydraulic platform for nuclear reactor simulation. It is jointly developed by CEA-DEN and EDF-DRD and also supported by IRSN and FRAMATOME-ANP. NEPTUNE is a new generation multi-scale platform. The system scale models the whole reactor circuit with 0D, 1D and 3D modules and is generally applied with a coarse meshing including about a thousand meshes. The component scale models components like the reactor Core or Steam Generators with a finer nodalization and is generally applied with 10^4 to 10^5 meshes. Since these components contain rod bundles or tube bundles the physical modelling uses a homogenization technique with a porosity. For some specific applications it was found necessary to add a two-phase CFD tool able to zoom on a portion of the circuit where small scale phenomena are of importance for design purpose or safety issues. Here the basic equations are still averaged like in RANS approach for single phase, but the space resolution is finer than in component codes and typical application may require 10^5 to 10^7 meshes. These three scales have to be coupled in order to simulate many reactor transients where both local effects and system effects play a role. In addition, two-phase Direct Numerical Simulation Tools with Interface Tracking Techniques can be used for even smaller scale investigations for a better understanding of basic physical processes and for developing closure relations for averaged models. The main challenges of this project are here presented and some first results are presented.

PHYSICAL VALIDATION ISSUE OF THE NEPTUNE TWO-PHASE MODELLING: VALIDATION PLAN TO BE ADOPTED, EXPERIMENTAL PROGRAMS TO BE SET UP AND ASSOCIATED INSTRUMENTATION TECHNIQUES DEVELOPED

Pierre Péturaud [1], Eric Hervieu [2]

[1] EDF R&D Division - 6 quai Watier - 78400 Chatou - France
pierre.peturaud @edf.fr

[2] CEA/Grenoble - DER/SSTH/LIEX - 38054 Grenoble cedex 9 - France
eric.hervieu@cea.fr

ABSTRACT

A long-term joint development program for the next generation of nuclear reactors simulation tools has been launched in 2001 by EDF (Electricité de France) and CEA (Commissariat à l'Energie Atomique). The NEPTUNE Project constitutes the Thermal-Hydraulics part of this comprehensive program. Along with the underway development of this new two-phase flow software platform, the physical validation of the involved modelling is a crucial issue, whatever the modelling scale is, and the present paper deals with this issue. After a brief recall about the NEPTUNE platform, the general validation strategy to be adopted is first of all clarified by means of three major features: *(i)* physical validation in close connection with the concerned industrial applications, *(ii)* involving (as far as possible) a two-step process successively focusing on dominant separate models and assessing the whole modelling capability, *(iii)* thanks to the use of relevant data with respect to the validation aims. Based on this general validation process, a four-step generic work approach has been defined; it includes: *(i)* a thorough analysis of the concerned industrial applications to identify the key physical phenomena involved and associated dominant basic models, *(ii)* an assessment of these models against the available validation pieces of information, to specify the additional validation needs and define dedicated validation plans, *(iii)* an inventory and assessment of existing validation data (with respect to the requirements specified in the previous task) to identify the actual needs for new validation data, *(iv)* the specification of the new experimental programs to be set up to provide the needed new data. This work approach has been applied to the NEPTUNE software, focusing on 8 high priority industrial applications, and it has resulted in the definition of *(i)* the validation plan and experimental programs to be set up for the open medium 3D modelling, successively in close connection with the in-PWR core DNB and PTS applications, and *(ii)* high priority experimental programs, with respect to porous medium multi-field and interfacial area transport, in connection with LB-LOCA application. These experimental programs will require the use of specific instrumentation to provide local characteristics of both liquid and vapour phases, such as local void fraction, local interfacial area concentration and local liquid velocity. To fulfil these needs, the following measurement techniques have respectively been developed/enhanced and assessed: X-ray tomography, 4-sensor optical probes and hot-film anemometry.

[2] Corresponding author

DIFFUSE INTERFACE METHODS FOR MULTIPHASE FLOW MODELING

Didier, Jamet[1]
DEN / DER / SSTH / LMDL – CEA/Grenoble – 38054 Grenoble Cedex 9
didier.jamet @cea.fr

ABSTRACT

Nuclear reactor safety programs need to get a better description of some stages of identified incident or accident scenarii. For some of them, such as the reflooding of the core or the dryout of fuel rods, the heat, momentum and mass transfers taking place at the scale of droplets or bubbles are part of the key physical phenomena for which a better description is needed. Experiments are difficult to perform at these very small scales and direct numerical simulations is viewed as a promising way to give new insight into these complex two-phase flows. This type of simulations requires numerical methods that are accurate, efficient and easy to run in three space dimensions and on parallel computers. Despite many years of development, direct numerical simulation of two-phase flows is still very challenging, mostly because it requires solving moving boundary problems. To avoid this major difficulty, a new class of numerical methods is arising, called diffuse interface methods. These methods are based on physical theories dating back to van der Waals and mostly used in materials science. In these methods, interfaces separating two phases are modeled as continuous transitions zones instead of surfaces of discontinuity. Since all the physical variables encounter possibly strong but nevertheless always continuous variations across the interfacial zones, these methods virtually eliminate the difficult moving boundary problem. We show that these methods lead to a single-phase like system of equations, which makes it easier to code in 3D and to make parallel compared to more classical methods. The first method presented is dedicated to liquid-vapor flows with phase-change. It is based on the van der Waals' theory of capillarity. This method has been used to study nucleate boiling of a pure fluid and of dilute binary mixtures. We discuss the importance of the choice and the meaning of the order parameter, *i.e.* a scalar which discriminates one phase from the other. In particular, we show the link between modeling and numerical issues related to the order parameter. We then present two new models, one dedicated to two-phase flows of non-miscible fluids and the other dedicated to liquid-vapor flows with phase-change. We show that these methods can deal with complex two-phase flows and that the well-posedness of these methods can give new insight into more classical methods based on the modeling of interfaces as surfaces of discontinuity.

[1] Corresponding author

TEST CASES FOR INTERFACE TRACKING METHODS: METHODOLOGY AND CURRENT STATUS

Olivier Lebaigue[1], Didier Jamet
CEA-Grenoble, DER/SSTH/LMDL, F-38054 Grenoble Cédex 9, France
olivier.lebaigue@cea.fr, didier.jamet@cea.fr

Hervé Lemonnier
CEA-Grenoble, DER/SSTH/LIEX, F-38054 Grenoble Cédex 9, France
herve.lemonnier@cea.fr

ABSTRACT

In the past decade, a large number of new methods have been developed to deal with interfaces in the numerical simulation of two-phase flows. We have collected a set of 36 test cases, which can be seen as a tool to help engineers and researchers selecting the most appropriate method(s) for their specific fields of application. This set can be use:

- To perform an initial evaluation of the capabilities of available methods with regard to the specificity of the final application and the most important features to be recovered from the simulation.
- To measure the maximum mesh size to be used for a given physical problem in order to obtain an accurate enough solution.
- To assess and quantify the performances of a selected method equipped with its set of physical models. The computation of a well-documented test case allows estimating the error due to the numerical technique by comparison with reference solutions. This process is compulsory to gain confidence and credibility on the prediction capabilities of a numerical method and its physical models.
- To broaden the capabilities of a given numerical technique. The test cases may be used to identify the need for improvement of the overall numerical scheme or to determine the physical part ofthe model, which is responsible for the observed limitations.

Each test case falls within one of the following categories:

- Analytical solutions of well-known sets of equations corresponding to simple geometrical situations.
- Reference numerical solutions of moderately complex problems, produced by accurate methods (*e.g.*, boundary Fitted coordinate method) on refined meshes.
- Separate effects analytical experiments.

The presentation will suggest how to use the test cases for assessing the physical models and the numerical methods. The expected fallout of using test cases is indeed on the one hand to identify the merits of existing methods and on the other hand to orient further research towards the mitigation of identified shortcomings. Moreover, it will also suggest a collaborative research effort on interface tracking methods which could be initiated by the direct use of the presented test cases and further developed if interest in such an activity is identified. The *Société Hydrotechnique de France* could provide a possible frame to host a working group.

[1] Corresponding author

177ème Session du comité scientifique et technique de la Société Hydrotechnique de France
Progrès récents des méthodologies de modélisation des écoulements diphasiques, Lyon, France, 24-26 novembre 2004.
Advances in the modelling methodologies of two-phase flows, Lyons, France, November 24-26, 2004.
Abstract number: 24

DEMONSTRATION OF A 3D PARALLEL IMPLEMENTATION OF THE FRONT-TRACKING METHOD: SIMULATION OF FLUID MIXING WITH A MOVING BOUNDARY AND A FREE SURFACE

Benoit Mathieu
CEA/Grenoble, DER/SSTH, 38054 Grenoble Cedex 9, France
benoit.mathieu@cea.fr

ABSTRACT

This paper presents an implementation of a front-tracking method for the numerical simulation of incompressible flows with moving boundaries or interfaces. The front-tracking method is based on a moving surface mesh that represents either an interface between fluid phases or an interface between the fluid and a solid. Contrary to the VOF method or the level-set method, the front-tracking method provides an explicit representation of the interface. This unique feature allows to describe the motion of the interfaces with great accuracy. Though, due to the complexity of the geometry handling algorithms, very few implementations of this method are sufficiently complete and robust to be used in an industrial environment.

The Trio_U code which is developed at CEA now features a parallel 3D front-tracking module that allows multiple interfaces to be tracked in the fluid. The coupling between the motion of the interfaces and the fluid allows to represent either multiphase flows including the effect of surface tension, and/or problems involving moving boundaries. Thanks to the object-oriented architecture of the code, it is possible to build a custom model within a few days. Hence, we present a simulation of mixing in an agitated vessel with a free surface. The agitator as well as the free surface are taken into account with the front-tracking method.

THE DYNAMICS OF DROPLETS ON CHEMICALLY AND TOPOLOGICALLY PATTERNED SUBSTRATES

A. Dupuis[1] and J. M.Yeomans
The Rudolf Peierls Centre for Theoretical Physics, 1 Keble Road, Oxford, OX1 3NP, England.
dupuis@thphys.ox.ac.uk, yeomans@thphys.ox.ac.uk

ABSTRACT

We develop lattice Boltzmann simulations to investigate the dynamics of micron-scale droplets spreading on chemically and topologically patterned substrates. For a substrate with lyophobic and lyophilic stripes of the same dimensions as the drop the final droplet shape is determined by the dynamic evolution of the fluid in a way which depends on the initial position and velocity. We compare the simulations to experiments and discuss how far a quantitative match can be made. We also consider droplets on a substrate covered by an array of micron-sized posts, a geometry which leads to superhydrophobic behaviour. We describe the dynamics of the transition from drops suspended on top of the posts towards drops collapsed in the grooves. Finally we show how the velocity of drops moving across the posts is related to the underlying geometry. The results are relevant to attempts to improve the quality of ink-jet printing.

[1] Corresponding author

LEDA, A NEXT GENERATION TOOL FOR PREDICTION OF MULTIPHASE FLOWS IN OIL AND GAS PIPELINES

Kjell Arne Jacobsen[1], Trondheim, Norway
SINTEF Petroleum Reasearch
Kjell.A.Jacobsen@iku.sintef.no

Stein Tore Johansen
SINTEF Materials and Chemistry, Trondheim, Norway
Stein.T.Johansen@sintef.no

ABSTRACT

Leda is a multiphase flow tool that has been developed over the last 3 years. This is a result of the challenges oil companies face when the continuously moving their exploration into deeper waters. Hence, the ambition is to develop technologies that can supply more accurate predictions that existing tools. This goal will be achieved by developing more physically based models, and by combining 1D models with 2D and 3D multiphase models when appropriate. The developed models are targeting all flow situations that may appear in single pipelines and pipeline networks. Based on fundamental multiphase physics, the models will not need flow regime maps and flow regime will be predicted.
The numerical technologies are targeted for N-phase multi component flows. The models developed are dealing with multi fluid and multi component tracking where phase transition is predicted as a result of local heat and mass transfer.
The paper will give an overall view of the LEDA project and show some temoraty results that have been produced.

[1] Corresponding author

IMPLICIT PENALTY METHODS ON EULERIAN GRIDS FOR THE SIMULATION OF INCOMPRESSIBLE MULTIPHASE FLOWS

Stephane Vincent, Jean-Paul Caltagirone, Stephanie Delage
Delphine Lacanette, Pierre Lubin, Tseheno Nirina Randrianarivelo
Laboratoire TREFLE, UMR 8508, Site ENSCPB
16, avenue Pey-Berland, 33607 Pessac Cedex, France

ABSTRACT

The numerical modelling strategy developped in our home made code Aquilon on Eulerian fixed grids will be presented paying attention to the implicit treatment of the conservation equations, the penalty methods used to solve various constrainsts, the algorithms for interface tracking (VOF, Front Tracking) and the automatic and adaptative methods for refining meshes (AMR). Detailed validations will be proposed to hiligth the convergence order and the consistency of the numerical methods and two- and three-dimensional physical validations will be leaded with comparisons to experimental or theoretical results. Finally, several examples of real two-phase flow simulations will be given such as fluidized bed motions, wave breaking on beaches, film wipping under turbulent jet action or mould filling processes. The results will be discussed with respect to experimental or industrial references and criticized for example accorting to the modelling of turbulence and its coupling with free surface.

MODELISATION NUMERIQUE DES ECOULEMENTS MIXTES DANS LES AMENAGEMENTS HYDRAULIQUES

Laurent Del Gatto[1]
Electricité de France, Centre d'Ingénierie Hydraulique
Savoie Technolac, 73 373 Le Bourget du Lac Cedex, France
laurent.del-gatto@edf.fr

Christian Bourdarias
Laboratoire de Mathématiques, UMR CNRS 8127
Université de Savoie, 73376 Le Bourget du Lac Cedex
bourdarias@univ-savoie.fr

RESUME

Les écoulements mixtes, définis comme des écoulements tantôt en charge, tantôt à surface libre, peuvent se présenter dans les galeries et conduites des aménagements hydrauliques, provoquant des fluctuations de pression importantes. Le Projet ROEMIX, collaboration entre EDF-CIH et l'Université de Savoie, a pour objectif le développement d'un code de simulation numérique 1D traitant des écoulements mixtes. Cette collaboration a conduit à l'écriture d'un système d'équations unique, représentant aussi bien les écoulements en charge que les écoulements à surface libre, et au développement du code de calcul ROEMIX, résolvant ce système d'équations, en régime transitoire, à l'aide d'un schéma numérique de type « volumes finis ».

NUMERICAL MODELLING OF THE MIXED FLOWS IN HYDRAULIC INSTALLATIONS

ABSTRACT

The mixed flows, definite as flows sometimes in pressure, sometimes in free surface, can arise in the galleries and conduits of hydraulic installations, causing important fluctuations in pressure. The ROEMIX Project, collaboration between EDF-CIH and University of Savoie, aims at the development of a 1D numerical code treating mixed flows. This collaboration led to the writing of a single system of equations, representing as well the pressure flows as the free surface flows. The computer code ROEMIX solves this system of equations, in transient state, using a numerical scheme of « finite volumes » type.

[1] Corresponding author

LEVEL-SET SIMULATIONS OF FLOWS WITH MOVING CONTACT LINES

Peter D. M. Spelt
Theoretical Mechanics, University of Nottingham, Nottingham NG7 2RD, U.K.
peter.spelt@nottingham.ac.uk

ABSTRACT

This work addresses the need for an accurate numerical technique for the simulation of two-phase flows that involve capillary motion, i.e., moving contact lines. Such flows occur in a variety of applications, from secondary oil recovery, mould-filling and coating processes to cell adhesion to blood vessels in shearing blood flow. Amongst the most popular numerical methods, level-set methods have been highly successful for the simulation of a wide variety of two-phase flows. Usually, these methods are used for cases in which there are no contact lines (such as surface waves). In this work, we investigate whether level-set methods can be used for accurate simulations of capillary flows as well. The main problem with such flows is that the motion of contact lines is determined to a large degree by details of the flow field in a tiny region around the contact line, such that extreme local mesh refinement would be required, in principle. We suggest here a possibility of resolving the large-scale motion only (such that no mesh refinement is needed). Results are presented for spreading droplets, and a comparison is made with analytical work and previous BEM simulations. Preliminary results are presented for the critical conditions beyond which a droplet or bubble can be displaced from an adhering wall by shear flow.

USE OF CFD FOR APPLICATIONS IN THE OIL AND GAS INDUSTRY

Ludovic Raynal
Process Division, Institut Français du Pétrole, BP 3, 69390 Vernaison – France
Phone: +33 (0) 4.78.02.25.27, Fax: +33 (0) 4.78.02.20.09, Email: Ludovic.Raynal@ifp.fr

ABSTRACT

CFD is more and more used in chemical engineering for application in the oil and gas industry. Its application field goes from calculations as an help for the tests of technological concepts to more fundamental work such as two-phase flow calculations in industrial reactors as a complementary tool to pilot plant tests. In this paper, two cases are reviewed : two-phase flow in bubble columns using Euler/Euler modelisation and two-phase flow in packed columns using the VOF approach. The contribution of CFD calculations is discussed for each case in terms of advantages and disadvantages in comparison with experiments. Limitations of present commercial CFD codes are discussed, some research needs in terms of model development are pointed out.