日本原子力学会　熱流動部会　　AESJ Thermal Hydraulics Division

THERMAL HYDRAULICS

熱流動部会ニュースレター（第 72 号）

AESJ-THD　NEWSLETTER (No.72)　Apr. 12, 2011

東北地方太平洋沖地震に際して

平成 22 年度部会長　大塚雅哉

平成23年3月11日の東北地方太平洋沖地震により被災された方々には心よりお見舞い申し上げます。また、福島第一原子力発電所の事態収束のため、現在も様々な形で対応、ご尽力いただいている部会員の方々も少なくないと思います。皆様のご努力に対して深く敬意を表します。

今回の福島第一原子力発電所での深刻な事態は、予想をはるかに超える津波によって引き起こされました。詳細な事象の進展は、今後検証を進めていく中で次第に明らかになってくると考えますが、主な要因は、津波により「冷やす」機能が喪失したことにあります。原子炉を守る冷却システムの健全性が保たれなかった事実を重く受け止め、再発を防止するとともに、今後に生かしていくことが必要です。熱流動部会として議論を深め、研究の方向性を確認し、提言していくことが求められます。部会員の方々には、それぞれの立場で今回の事象を見つめ、原子力安全の強化、技術向上に向けた取り組みに結び付けていただくようお願い申し上げます。

福島第一原子力発電所の対策では、事態収束のため、現在も懸命な取り組みが続けられています。引き続き、関係されている方々への直接的、間接的な支援を宜しくお願い致します。

研究室紹介

Research Activities on the Use of CFD Codes for Thermal Hydraulic at the Chair of Nuclear Engineering of the Technische Universität München

Department of Nuclear Engineering Technische Universität München
Rafael Macián-Juan and Filippo Pellacani

I) INTRODUCTION

The Chair of Nuclear Engineering (NTech) was established in April 2007 as part of the Mechanical Engineering Faculty of the Technische Universität München (TUM), partially supported by an endowment from E.ON Nuclear Power. The academic and research activities of NTech have been guided by three main goals: (i) to carry out research in nuclear engineering at an international level by using, improving on and developing advanced methodologies for nuclear safety applications; (ii) to create an attractive academic program for students interested in the field of nuclear technology, and (iii) to establish a network of national and international contacts that can support research and academics in nuclear engineering at TUM.

To fulfill the academic goals, we have developed a curriculum, based on Bachelor and Master degrees in Nuclear Engineering, that can offer interested students fundamental and advanced knowledge on nuclear engineering, thus preparing them for employment in the nuclear industry, nuclear research organizations, or nuclear regulatory authorities after graduation. The curriculum includes introductory courses designed to attract a wider spectrum of students by exposing them, for the first time, to nuclear technology. More advanced and specialized courses, and an agreement for academic cooperation with the Institut National des Sciences et Techniques Nucleaires (INSTN) at Saclay, France, allows students interested in developing a nuclear engineering career to pursue the bachelor and master degrees.

Our research activities focus on the use of state-of-the-art computer based analytical tools and methodologies for the safety evaluation of current and future nuclear systems. This use is supported by the assessment, improvement, and development of physical and numerical models implemented in these codes and by the development of methodologies that make use of them in an integrated framework (multi-physics coupling) in order to better address the safety of operation and the design concerns of a variety of nuclear reactor designs, especially of the Light Water Reactors (LWRs) type.

The research activities are based on the following lines:

•Thermal-hydraulic and Neutronic coupled nuclear systems analysis.

•Analytical Thermal-hydraulics with the use of Computational Fluid Dynamics (CFD) for nuclear safety analyses.

•Uncertainty Analysis applied to nuclear safety: the development and application of statistical methodologies able to quantify the uncertainty in the result of nuclear code simulations.

•Nuclear Dynamics: transient and stability behavior of BWRs and PWRs.

•Stochastic and Deterministic Radiation Transport for criticality and neutron fluence analysis and for medical applications of radiation and radiation safety (dose determination.)

A complementary experimental research activity involves thermal-hydraulic experiments in a low pressure experimental facility, currently in the building and commissioning phase. It has been designed to carry out experimental measurements on two-phase flow phenomena such as condensation of bubbles in sub-cooled vertical flow and horizontal interfacial drag in co-current and counter-current flow at low pressure.

Research activities are carried out in collaboration with several national and international groups such as GRS, TÜV-SÜD, Forschungszentrum Dresden (FZD), Karlsruhe Institute of Technology (KIT), ANSYS-CFX in Germany, Polytechnic University of Valecia and University Jaume I in Spain, etc.

This paper will briefly describe the research line of application of CFD codes to nuclear safety research.

II)CDF APPLICATIONS TO NUCLEAR SAFETY ANALYSIS

The objective guiding the work on the application of CFD codes to safety analysis is to assess their adequacy and to improve their simulation capabilities so that they can become reliable analytical tools for design and safety related studies of current and future nuclear reactors. The aim is toachieve flow descriptions in nuclear systems with an accuracy and local resolution beyond those that the results of current state-of-the-art Systems Analysis Codes (e.g. ATHLET, RELAP-5, TRACE) and sub-channel analysis codes (e.g. COBRA-TF) can offer. This is especially important in the case of simulations of local thermal-hydraulic effects whose influence on the fuel rod neutronic response and thermal behavior will have a significant inpact on fuel design performance and on reactor safety. CFD codes are expected to significantly increase the accuracy of the numerical predictions once their physical and numerical models are computationally robust enough and capable of dealing with the flow conditions expected during normal and off-normal operation of nuclear reactors.

In this context we are following an strategy that pursues research in three main areas: (i) accurate modeling of single phase nuclear fuel assemblies: addressing turbulence effects, calculation of heat transfer coefficients, and prediction of boiling phenomena; (ii) the improvement of the two-phase CFD modeling capability of current CFD tools for interfacial heat, mass and momentum (friction) transfer; and (iii) The coupling of the thermal-hydraulic CFD flow description to neutronic core models (diffusion and neutron transport) at the level of nuclear fuel assemblies with single fuel rod resolution, which could initiate the way towards future transient analysis of a complete nuclear core with individual fuel rod accuracy.

The two main codes used are ANSYS-CFX and OpenFOAM, which are run in parallel LINUX and Windows computing environments on our multiprocessor clusters and on the massively parallel German Leibniz-Supercomputing Center (LRZ).

The work produced so far has involved a series of PhD projects and Master theses addressing the development of bubble transport and boiling models for poly-dispersed bubbly flows; modeling of direct contact condensation; and modeling of interfacial drag and interfacial area transport for low void fraction flows. On the other hand, complementing this work, the determination of the two-phase flow simulation capabilities of current CFD codes has focused on the assessment of thermal (energy) stratification and the effects of turbulent mixing, condensation heat transfer and bubble transport for bubbly flow, and boiling with and without energy and mass phase interchange. A summary of some of the results of the latter activity, carried out in collaboration with Sylvana Matturro and Prof. Sergio Chiva at the University Jaume I in Spain is the main technical subject of this paper presneted in teh following sections.

III)MATEMATICAL MODELS

1. Two-fluid model

Momentum Conservation The simulations presented in this paper are based on the two-fluid model Eulerian–Eulerian approach. The liquid phase is considered as the continuous phase and the gas phase is considered as dispersed. In the isothermal case no interfacial mass transfer takes place. The momentum equation for the two-phase mixture can be expressed as follows:

$$\frac{\partial\left(\rho_i\alpha_i\vec{U}_i\right)}{\partial t}+\nabla\cdot\left(\rho_i\alpha_i\vec{U}_i\vec{U}_i\right)= -\alpha_i\nabla P+\rho_i\alpha_i\vec{g}+\nabla\cdot\left[\mu_i\alpha_i\left(\nabla\vec{U}_i+\left(\nabla\vec{U}_i\right)^T\right)\right]+F_i \quad (1)$$

The term F_i in Eq. (1) represents the total interfacial force acting on the phases**.** Closure laws are needed to calculate the momentum transfer of the total interfacial force.

Modelling of the interfacial Forces Four interfacial forces have been considered during the analysis. The drag force F_D, has been modeled using the Grace model [1]. The non-drag forces considered are the lift force F_L, the wall lubrication force F_{WL}, and the turbulent dispersion force F_{TD}. The virtual mass force was neglected, since tests conducted by Frank et al. [2] showed that its influence is of minor importance in comparison with the amplitude of the other drag and non-drag forces.

$$F_i = F_D + F_L + F_{WL} + F_{TD} \quad (2)$$

Each of the forces needs empirical closure relations to determine the various coefficients appearing in their formulations.

Drag Force The drag force accounts for the drag of one phase on the other and the coefficient that has been used for this force is that of Grace et al. [1].

Lift Force Due to velocity gradients, bubbles rising in the liquid bulk are subjected to a lateral lift force. This is modeled according to the Tomiyama [3] formulation

$$\vec{F}_L = -C_L.r_\beta\rho_\alpha.(\vec{U}_\beta-\vec{U}_\alpha).\overrightarrow{rot}\,(\vec{U}_\alpha) \quad (3)$$

For the evaluation of the lift coefficient C_L two methods have been used, namely, a constant value of 0.06 (a low value still in the range reported by the ANSYS CFX software manual) and a value calculated according to Tomiyama [2].

Wall Lubrication Force Due to surface tension, a lateral force appears to prevent bubbles from attaching to the solid wall. The wall lubrication force has been modeled as follows:

$$\mathrm{F}_{\mathrm{WL}}^{\mathrm{fluid}} = -\mathrm{F}_{\mathrm{WL}}^{\mathrm{gas}} = -\alpha_{\mathrm{d}}.\rho_{\mathrm{c}}.\frac{1}{\mathrm{d_p}}.\mathrm{C_{WL}}.\left(\vec{\mathrm{U}}_{\mathrm{rel}}-\left(\vec{\mathrm{U}}_{\mathrm{rel}}.\hat{\mathrm{n}}_{\mathrm{w}}\right).\hat{\mathrm{n}}_{\mathrm{w}}\right)^2.\hat{\mathrm{n}}_{\mathrm{w}} \quad (4)$$

The two lubrication force coefficients are adjusted in order to simulate these two effects: to achieve a higher absolute value of the wall lubrication force and also to extend its action so that it has an effect far from the near wall region.

The implementation of the wall lubrication force is necessary for adiabatic two-phase flows, as it reproduces the void fraction peak near the wall well [4]. Krepper [5] reports that its use in case of high-pressure, wall boiling conditions may be questionable, but also that further research is necessary to improve the existing wall force models.

Turbulent dispersion force and bubble induced turbulence A turbulent dispersion force has been considered to take into account the turbulence assisted bubble dispersion. The turbulent dispersion force model that has been used is the Favre averaged Drag force (FAD) [6]. The Bubble induced turbulence has been taken into account according to Sato's [7] model.

2. Modeling of the wall boiling

The wall partitioning model proposed in the work of Kurul and Podowski [8] is based on the division of the heat flux applied on the heated surface into three different terms, convective, quenching and evaporative.

$$Q_w = Q_e + Q_c + Q_q \quad (5)$$

For the definition of each term closure relationships are required. In the actual state of development they are based essentially on empirical correlations rather than on physical, mechanistic models.

IV)VALIDATION OF THE MODELS

To show the adequacy of the calculated results, the interfacial force models and the boiling model, described previously were implemented in ANSYS-CFX to simulate, for the adiabatic case, the experiment of Hibiki 2001 [9] and for the non-isothermal case, the experiment of Bartolomej 1967 [10].

1. ANSYS CFX calculation grid

Several grids were tested in order to determine an appropriate computational domain for the conditions of the simulations. The mesh sensitivity analysis yielded the shortest computational time and the independence of the results from the calculation grid for mesh composed of ca. 110000 nodes for the isothermal case and 70000 for the subcooled boiling case. The CFD model represents one eighth of a vertical pipe using symmetric boundary conditions for both axial cut planes. The first node near the wall was set at a distance yielding a value of y^+ around 30÷40 for the isothermal case and 60÷70 for subcooled boiling case, in order to avoid numerical oscillations and to result in an accurate wall lubrication force modeling. The

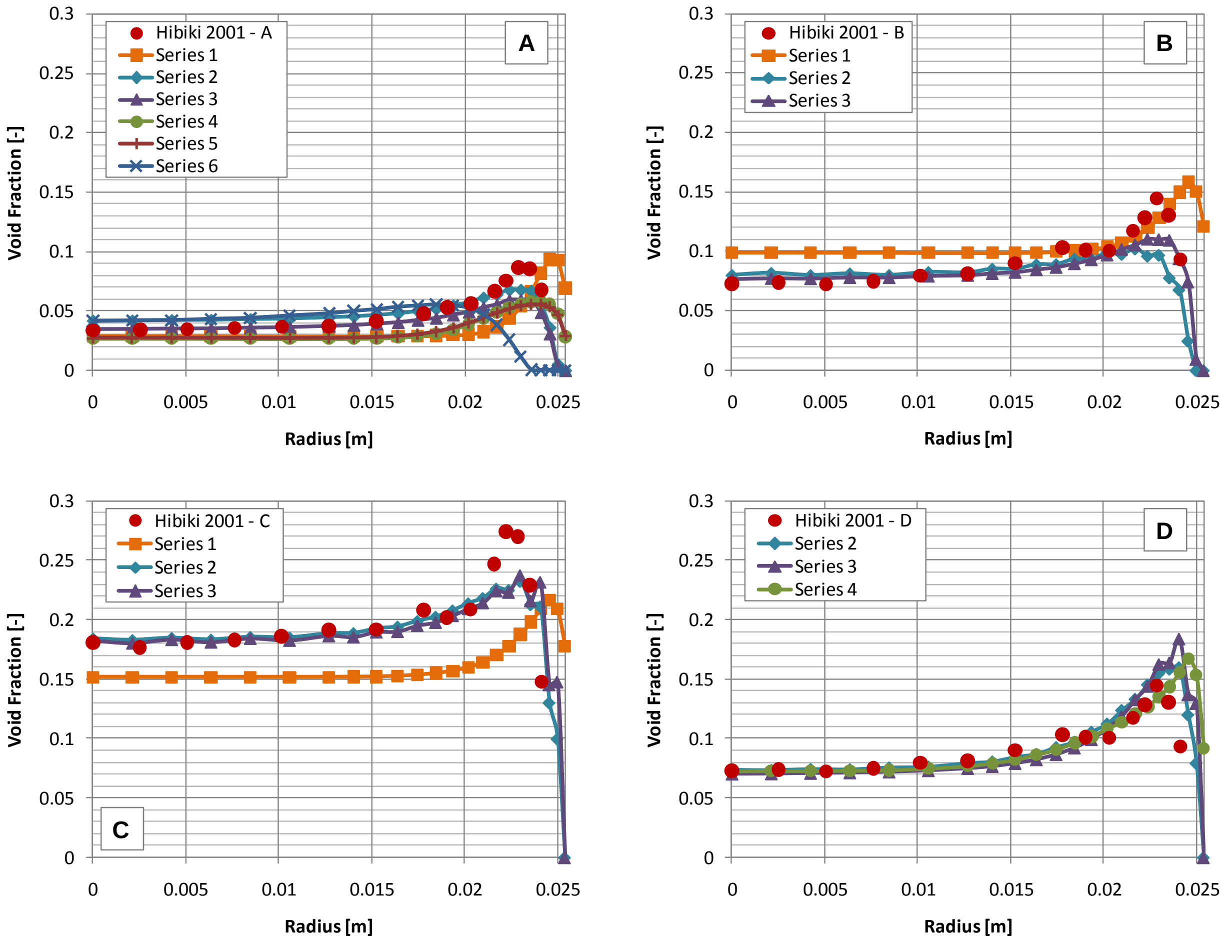

Fig.1 Dependence of α on r and j_g.

bottom boundary conditions were extracted from the experimental data reported in the papers describing the measurements. At the outlet section a constant pressure boundary condition at atmospheric pressure was considered. A RANS turbulence model, based on the SST, for the liquid phase was selected and, for the subcooled boiling case, also the k-ε model was tested. A zero equation turbulence model desccribed the gas phase. The lift force was modeled using the Tomiyama model [11] and the Grace model [1] model modelled the interfacial drag force.

2. Hibiki2001 - Calculation results

In all the simulations the wall lubrication force was based on the Antal formulation [2], with coefficients from Krepper et al. [12], and Frank et al. [2] (see Table 1 for a detailed overview). For this case not only the original coefficients [2] have been used, but other two sets of coefficients were also tested with the aim of reducing the high absolute value of the wall lubrication force predicted with the standard values and also to restrict its influence on the flow to only the near wall region.

In was observed that the Tomiyama lift force model as modified by Frank et al. [2] with the original coefficients led to very high values for this force when the distance of the first computationsl node from the wall is very small. This can lead to numerical instabilities during the calculation and an increased value of y^+ was then selected. For experimental cases A and D, also the poly-dispersed MUSIG approach was been tested (Series 4 and 5).

Table.1 Parameters of the Wall lubrication force tested in the simulations

Series	WLF	C_1	C_2	-	MUSIG
1	Antal	-0.0064	0.016	-	NO
5	Antal	-0.0064	0.016	-	YES
Series	-	Cwc	Cwd	p	MUSIG
2	Frank	8	8	1.2	NO
3	Frank	10	6.8	1.2	NO
4	Frank	8	8	1.2	YES
6	Frank	10	6.8	1.7	NO

According to Fig. 1 A,B,C - Series 1 the Antal model with coefficients from Krepper and Prasser [12] leads to and underestimation of the wall lubrication force effect. The void fraction distribution along the radius presents a peak that is not in accordance with the experimental data. Simulating the wall lubrication force using the model that was modified by Frank [2] with original coefficients (Fig. 1 A - Series 6) leads to the overestimation of the force effect. This trend is also shown in Figure 3 representing the C_{WL} as a function of the distance to the wall. The new sets of coefficients tested for the Tomiyama model modified by Frank [2], led to essentially similar results (Figure 1 A,B,C,D - Series 2 and 3). The position of the void fraction peak, in these cases, is in good agreement with the experimental profiles. The homogeneous polydispersed approach (Homogeneous MUSIG) also tested (Fig.1 A – Series 4 and 5; Fig. 1 D – Series 4) This approach led to results in line with the monodispersed approach (Series 2 and 3) but with a much larger computational effort.

3. Bartolomej 1967 - Calculation results

It is important to mention that the assessment presented in Ref.[5] for the same experimental data and computer code has several differences in modeling assumptions compared to our work. Krepper at al [5] selected the following closure models for the setup of the calculation:

The SST turbulence model for the liquid phase, the Sato model for the bubble induced turbulence, the Ishii–Zuber model of the interfacial drag. The only non-drag forces considered was the Favre-averaged turbulent dispersion force. This modeling strategy resulted in a good agreement with the experimental data over a wide range of flow conditions.

In our contribution a wider range of models concerning turbulence, drag and non drag forces models and coefficients has been tested.

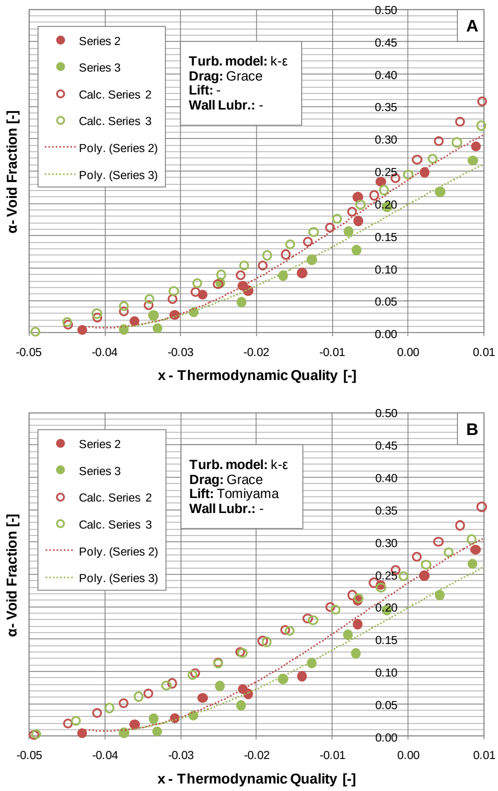

Fig.2 Dependence of α on x - effect of the lift force model of Tomiyama [3].

These modeling assumptions yield results that differ from those in Ref.[5] in some cases. However, it is not the purpose of this paper to explain these discrepancies in detail, but to show the influence of the proposed model assumptions in the accuracy of the simulations by comparing them to the experimental data.

Figure 2 shows the results obtained when the subcooled boiling is modeled with and without the presence of a lift force (figures 2B and 2A respectively).

In general the void fraction versus the thermodynamic quality is always over predicted by ANSYS CFX, but more when the lift model is considered. Since the governing system parameters (P, q'', G_{in}) allow the generation of bubbles that remain far below the critical value of 5.8 mm for the lift force; the lift coefficient has a constant positive value of 0.288 (in equation 5 we are in the range for the Eötvos number Eo_d<4). The bubble diameter was modeled according to Kurul and Podoswki [13], with the bubble size dependent on liquid subcooling. Exactly, the bubble diameter is inversely proportional to the liquid subcooling.

The lift force, with a high predicted value, prevents the bubbles generated at the wall from moving to the centerline (Fig. 2B) where they could condense since in this region the liquid is still subcooled. In fact, during subcooled boiling in the region where the thermodynamic quality is negative (x<0) steam condensation process takes placeand as bubbles are swept far from the wall by internal or external forces to zones where the temperature is lower than the saturation temperature, they collapse.

When saturation conditions are reached (x>0), the results of Fig. 2 (A) and (B) are comparable.

The lines labeled "*Poly.*" represent polynomial fits of the experimental results as a means of better assessing the quality of the simulation results.

In the simulations results shown in Fig. 3 the wall lubrication force model of Antal with the coefficients from Krepper et al. [12] was added. The wall lubrication force under these conditions has a very low effect on the flow and is concentrated in the wall near region were the temperature of the liquid is near of above saturation. The results are similar to the case of Fig. 2 (B).

In Fig. 4A, a constant lift coefficient was set with a value of 0.06 with the aim of reducing the effect of the lift force that prevented the bubble from moving towards the centerline. The Tomiyama coefficient used in the previous calculations (Fig. 2 (B) and Fig. 3) was ca. 5 times higher.

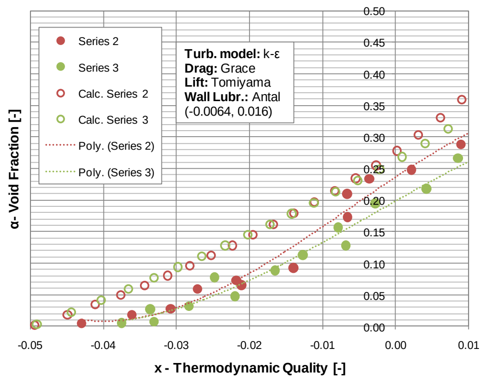

Fig.3 Dependence of α on x - effect of the wall lubrication force (Antal with coefficients from Krepper et al. [7]).

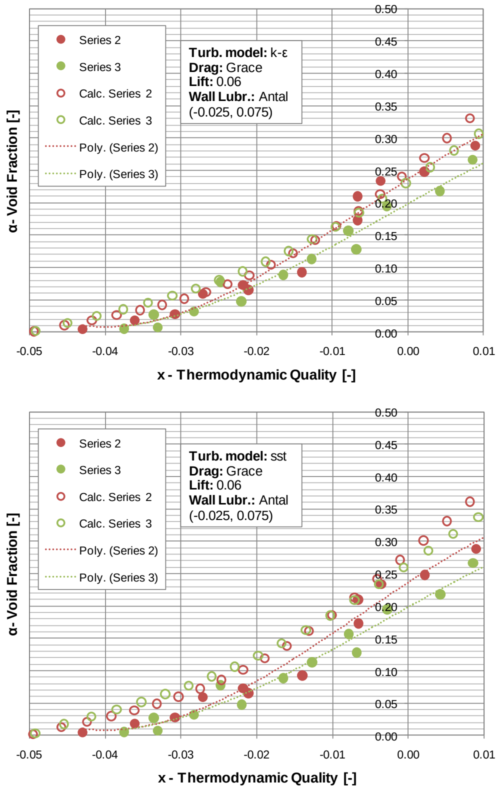

Fig.4 Dependence of α on x - (a) effect of the wall lubrication force, (b) effect of the sst turbulence model (Antal with coefficients C_1=-0.025 C_2=0.075).

The two lubrication force coefficients were adjusted in order to obtain two effects: to achieve a higher absolute value of the wall lubrication force and also to extend its action not only at the near wall region. This resulted in lowering the calculated axial mean value of the void fraction to the level of the experiments, even if the code still slightly over-predicted the void fcation.

The SST turbulence model was also tested (see Fig. 4b) and the effect on the results was the increase of the void fraction for the same thermodynamic quality. The SST model resulted in a better turbulent mixing of the flow which lowered the temperature difference between the wall and the flow centerline and reduced the collapse of the bubbles.

V)CONCLUSIONS

The assessment of models for the two-phase flow phenomena includes a staged approach. In the present work two test cases were analyzed with the specific goal of assessing the current models used for the forces, turbulence, bubble diameter and the inter-phase energy exchange in one of the state-of-the-art CFD two-phase flow capable codes, ANSYS CFX 12.

In general good qualitative agreement was obtained between the experimental data and the simulations results. The models give predictions in close agreement with experimental results.

In case of adiabatic bubbly flow the radial profiles of the void fraction obtained from the simulation are in good agreement with the experimental results produced by Hibiki et al. [9].

In case of subcooled boiling the agreement is good if the axial mean values are considered. The main difficulties for the simulation are observed for flow in transition to flow regimes with higher void fraction at the end of the subcooled boiling region. Here the bubbly flow is not able to maintain the spherical condition of the bubbles, which is a requirement of the boiling models.

The assessment of the interfacial force models used in this work has been carried out based on water air data at low pressure and room temperature. In the literature, experimental data at high pressure with detailed radial profiles for the most important physical parameter like phase velocities, temperatures, void fraction, are not easily found. For these reasons radial profiles for the above mentioned parameters in case of high pressure conditions are not shown in the present work. The enlargement of the experimental database for bubbly flow and subcooled boiling at high and low pressure with an adequate level of resolution is required for further development. Radial profile distributions of void fraction, gas and liquid velocities and liquid temperature will allow the comparison of calculation results with experimental data to assess and validate further models to enlarge the range of applicability of CFD codes in the field of the two-phase flow simulations.

VI)REFERENCES

[1] Grace J.R., Clift R., Weber M.E., Bubbles, Drops and Particles, Academic Press, 1978.

[2] Frank Th., Zwart P.J., Krepper E., Prasser H.-M., Lucas D., Validation of CFD models for mono- and polydisperse air–water two-phase flows in pipes, Nuclear Engineering and Design 238 (2008) 647–659.

[3] Tomiyama, A., Struggle with computational bubble dynamics, 3rd International Conference on Multiphase Flow, ICMF-2004, Lyon, France, June 8–12, 1998, pp. 1–18.

[4] Lucas, D., Shi, J.-M., Krepper, E., Prasser, H.-M., Models for the forces acting on bubbles in comparison with experimental data for vertical pipe flow. In: 3rd International Symposium on Two-Phase Flow Modelling and Experimentation, Pisa, Italy, 2004.

[5] Krepper E., Končar B., Egorov Y., CFD modelling of subcooled boiling—Concept, validation and application to fuel assembly design,Nuclear Engineering and Design 237 (2007) 716–731.

[6] Burns, A.D.B., Frank, Th., Hamill, I., and Shi, J-M., Drag Model for Turbulent Dispersion in Eulerian Multi-Phase Flows, 5th International Conference on Multiphase Flow, ICMF-2004, Yokohama, Japan.

[7] Sato, Y. and Sekoguchi, K.,Liquid Velocity Distribution in Two-Phase Bubbly Flow, Int. J. Multiphase Flow, 2, p.79, 1975.

[8] Kurul, N. and Podowski, M.Z., On the modeling of

multidimensional effects in boiling channels, ANS Proc. 27th National Heat Transfer Conference, Minneapolis, MN. 1991.

[9] Takashi Hibiki, Mamoru Ishii, Zheng Xiao, Axial interfacial area transport of vertical bubbly flows, International Journal of Heat and Mass Transfer, Volume 44, Issue 10, May 2001, Pages 1869-1888.

[10] Bartolomej, G.G., Chanturiya,V.M., Experimental study of true void fraction when boiling subcooled water in vertical tubes, Thermal Engineering, vol. 14, pp. 123–128 1967.

[11] Antal, S.P., Lahey, R.T., Flaherty, J.E. Analysis of Phase Distribution in Fully Developed Laminar Bubbly Two-Pahse Flow, Int. Journal of Multiphase Flow, Vol 17, 635-652, 1991.

[12] E. Krepper, Lucas D., Prasser H.-M., On the modelling of bubbly flow in vertical pipes, Nuclear Engineering and Design 235 (2005) 597–611.

[13] Kurul, N., Podowski, M.Z., Multi-dimensional effects in sub-cooled boiling. In: Proc. 9th Heat Transfer Conference, Jerusalem, 1990.

運営委員会報告

熱流動部会運営委員会(H22－2) 議事録

(1) 日時：平成 23 年 1 月 25 日(火) 13:00-17:00
(2) 場所：日本原子力学会　会議室
(3) 配布資料：
① 議事次第
② 総務委員会活動報告
③ 研究委員会報告
④ 国際委員会の活動概要
参考資料④－①　NTHAS6 予算案・サマースクール決算書
参考資料④－②　日韓原子力学生-若手研究者交流事業運営小委員会規約・メール審議内規・覚書
⑤ 広報委員会活動報告
参考資料⑤－①　熱流動部会ニュースレター(第 71 号)
⑥ 出版編集委員会の活動概要
⑦ 追加資料：ANS との連携

議事

1. 総務委員会報告（大川総務委員長）

1.1. 予算関連報告

平成 22 年度収支・決算（見込）および平成 23 年度予算申請状況ついて、資料②に基づき説明があり、了承された。また、ドクターフォーラム、NURETH、NUTHOS、NTHAS、日韓学生セミナーの開催状況を考慮して、平成 27 年度までの長期予算計画について説明がなされた。これに関連して、学会の一般法人化に向けた部会繰越金の取扱いについて情報交換を行った。

1.2. その他活動報告

資料②に基づき、総務委員会の活動報告が行われた。報告項目は以下の通り。春の年会企画セッション開催関連、部会等 WG のメンバー選出調整、部会賞準備関連、安全工学シンポジウム開催関連、春の年会プログラム編成関連、代議員推薦関連、フェロー推薦関連、原子力安全功労者推薦関連。なお、安全工学シンポジウムへの協力体制については、原子力安全部会と意見交換が必要との意見が出された。

1.3. 協議事項

春の年会におけるポスターセッション発表賞選考委員の推薦依頼について説明があり、本部会からは大塚部会長を推薦することとした。
規約類の公開方法について議論し、部会規約、役員任期規程、部会表彰規定について、部会規約の制定年月日の確認を行った後、部会 HP 上で公開することとした。

2. 企画委員会報告（木村企画委員長・代理：大野国際委員長）

秋季セミナー「Dr.フォーラム」について、平成 22 年度分の実施報告と平成 23 年度分の準備状況が報告された。23 年度は、秋の大会が祝日と NURETH-14 に挟まれる日程となるため、開催時期も含めて検討を実施中である旨が報告された。

3. 研究委員会活動報告（山本研究委員長）

資料③を用いて、「二相流データベースの整備・詳細評価研究専門委員会」および「熱水力安全評価基盤技術高度化検討 WG」の活動状況報告があった。特に、二相流データベースの整備・詳細評価研究専門委員会については、設置期間が今年度末までとなるため、報告書および今後に向けての提言を策定中である旨が報告された。また、春の年会における企画セッションのテーマの紹介があった。

4. 国際委員会活動報告（大野国際委員長、池田国際副委員長）

資料④に基づき、NTHAS7 及び日韓学生セミナーの実施報告と、NURETH-14 及び NTHAS8 の準備状況報告がなされた。NTHAS8 については、山口彰教授（阪大）を中心に開催地と開催時期を検討中であり、現時点での候補が示された。また、組織委員会の候補者選出方針について意見交換を行った。

5. 広報委員会活動報告（玉井広報委員長、染矢広報副委員長）

資料⑤に基づき、部会 HP の更新状況、ニュースレターの発行状況、メーリングリストによる情報提供状況について報告が行われた。ニュースレターの記事として、ANS 熱流動部会の活動内容を紹介するなどのアイディアが示された。また、染矢広報副委員長より、ニュースレター71 号のドラフトが提示された。

6. 出版編集委員会活動報告（宋出版編集委員長）

資料⑥に基づき、H18～H22 年度現在までの投稿論文数の推移が説明された。また、学会の編集委員会におきて、Elsevier や Taylor and Francis 等の大手出版社への論文公開委託について検討が行われている旨、紹介があった。

7. その他

二ノ方海外担当役員より、今後の ANS との連携について説明があった。特に、NUTHOS の開催に関しては、日中韓、台湾等との連携および運営方法について今後議論を進めていく必要性が示された。

以上

平成 22 年度　熱流動部会役員

部会長	大塚　雅哉	（日立）
副部会長	片岡　勲	（大阪大学）
総務委員長**	大川　富雄	（大阪大学）
総務副委員長**	西　義久	（電力中央研究所）
広報委員長**	玉井　秀定	（JAEA）
同副委員長*	染矢　聡	（産業技術総合研究所）
研究委員長**	山本　泰	（東芝）
国際委員長**	大野　修司	（JAEA）
同副委員長*	池田　秀晃	（三菱重工）
企画委員長*	木村　暢之	（JAEA）
出版編集委員長**	宋　明良	（神戸大学）
同副委員長*	波津久　達也	（東京海洋大学）
表彰委員長	秋本　肇	（JAEA）
海外担当役員	二ノ方　壽	（東京工業大学）

*:任期2年の1年目、**:任期2年の2年目

国際会議カレンダー（Web のみに掲載）

熱流動部会のホームページ http://wwwsoc.nii.ac.jp/aesj/division/thd/ より最新の情報を入手して下さい。

＜編集後記＞

ニュースレターへの原稿は、随時受付を行っております。研究室紹介、会議案内、エッセイ等寄稿お願い致します。72号でも70、71号に引き続き，国外の研究室を紹介致しました。またニュースレターに関するご質問、ご意見、ご要望等ありましたら、ぜひ下記宛にe-mailをいただければ幸いです。熱流動部会に入会したい方、入会しているがメールが届かない方が身近におられましたらご相談ください。

e-mail宛先： tamai.hidesada@jaea.go.jp
s.someya＠aist.go.jp

熱流動部会のホームページ：
http://wwwsoc.nii.ac.jp/aesj/division/thd/
からニュースレターの PDF ファイルは入手可能です。