PHOENICS NEWS

Editorial

Summer 2012

CHAM will soon release PHOENICS 2012 which includes the following:

New WRITE-file feature of Satellite which enables Q1 files to create batch files, PHOTON macros and object-creation files.

New EXEC feature of Satellite which enables Q1 files to run batch files and executables such as ShapeMaker and AC3D.

Improved ShapeMaker which accepts instructions via files and creates never-before-existing objects with parameters specified by the user.

New Datmaker which:

- repairs CFD-deficient CAD files;
- recognises as distinct the numerous separate objects sometimes represented in a single CAD file;
- converts separate but touching single objects consisting of a single material into a single object;
- does the same for interpenetrating objects;

These attributes simplify the solver-module's task of recognising how object surfaces cut cell edges and so reduces its errors.

Much improved treatment of cut-cell edges when the grid is polar.

Speeded-up, parallelised GCV solver recommended for all body-fitted-co-ordinate problems.

Improvements to the unstructured-grid capability which are specific to the wind-farm-simulation problem.

Various bug fixes.

If you are interested in further details, or in obtaining PHOENICS please contact sales@cham.co.uk. If you are a maintained PHOENICS User and would like to receive PHOENICS 2012 please contact Michelle Lyle (mjl@cham.co.uk) to arrange delivery.

2) PHOENICS News

2.1 Branch and Agent News

2.1.1 Benelux PHOENICS User Meeting arranged by A2Te

The 2012 Benelux PHOENICS User Meeting (BPUM) was held in Eindhoven, Netherlands. Attendees met fellow users, discussed their experiences, learned new ways of working at the workshop and heard about the latest developments. Dr Geert Janssen was present from A2Te and Dr David Glynn from CHAM.

CHAM's Dr David Glynn outlines some of the new features of PHOENICS/FLAIR to some of the participants at the 2012 Benelux User Meeting held in Eindhoven.

2.1.2 Focus Advanced Technologies Sdn Bhd

CHAM is pleased to announce the appointment of **Focus Advanced Technologies** as its Agent in Malaysia.
Focus staff have CFD capabilities and particular expertise in:

- Heat Modelling in Fire and Smoke in Building
- Sport Science
- Biomedical Application
- Automotive Aerodynamics

Support and Training: Focus Advanced Technologies will provide training for PHOENICS Users at three levels:

- beginner: specifically for first time users with little or no background in engineering;
- intermediate: for those with an engineering background who want to add new skills;
- advanced: for those who use CFD regularly but require deeper understanding on the theory behind the CFD software.

Contact: Focus Advanced Technologies Sdn Bhd, C-2-16, 2nd Floor, Block C, Megan PHOENICS, Jalan 2/142A, Off Jalan Cheras, 56000 Kuala Lumpur, Malaysia
www.focus-technologies.com.my,
e-mail: fadil@focus-technologies.com.my, kamarul@focus-technologies.com.my
tel: 03-91022377 (Office), fax: 03 – 91002477,

2.1.2 News from Shanghai Feiyi

Shanghai Feiyi has become a member of Zhejiang Province's Civil Building Energy Saving Evaluation Association and PHOENICS has been chosen to be the main evaluation software for Venting and Heat Island analyses in the province of Zhejiang.
On the last Friday of every month, training courses and user meetings are held at the offices of Shanghai Feiyi.
Shanghai Feiyi is offering an online support system so that maintained PHOENICS Users can get timely user support.

2.2 PHOENICS Activities

Dates	Activity
September 9	*PHOENICS at Semi-Conductor & Processing Equipment Conference R.O.C. Contact: sales@c-h-a-m-p-i-o-n.com.tw*
September 10 – 11	*BSO12 Conference, CHAM is sponsoring "Best Paper" at the Building Simulation and Optimisation 2012 conference to be held at Loughborough University. See **http://www.bs012.org/** for details*
September 17 – 18	*Advanced Training (Fire, Smoke and Safety in Buildings) at CHAMPION Contact: sales@c-h-a-m-p-i-o-n.com.tw*
September 18 – 20	*PHOENICS/ FLAIR Training Course This is one of CHAM's regular courses and comprises a three-day introductory course. Attendees are welcome to come for only part of the time; email sales@cham.co.uk to register for this course or obtain further information. Courses can also be run at customer sites by prior arrangement; email as above for information.*
October 15 - 17	*Basic Training at CHAMPION. Contact: sales@c-h-a-m-p-i-o-n.com.tw*
November 12 - 13	*Advanced Training (CFD in Semi Conductors & Opteoelectronic Processing Equipment) Contac:t sales@c-h-a-m-p-i-o-n.com.tw*
November 13 - 15	*PHOENICS/ FLAIR Training Course This is one of CHAM's regular courses and comprises a three-day introductory course (see description above).*
November 16 – 17	*Demonstration & Presentation at the National Conference of Theoretical and Applied Mechanics, R.O.C. Contact sales@c-h-a-m-p-i-o-n.com.tw*

3) PHOENICS Applications

3.1 CFD Modelling of an Artificial Reef by

Rama devi Pathakota & Paul Emmerson, CHAM Limited

Introduction: Artificial reefs are man-made steel structures which are deployed on the seabed, typically built in areas with a generally-featureless bottom to promote marine life, control erosion, block the passage of ships, or improve conditions for surfing. These structures, when placed on the seabed, tend to increase the local current and wave motions. For a sand/clay/silt type of seabed, fast flowing water can stir up fine particles, picking them up and transporting them away from the structure, creating a hole. This phenomenon is called scouring, which can weaken the support structure, possibly causing it to fall over into the resulting pit. A most cost-effective and fastest method for evaluating the initiation of scouring on the seabed is through CFD simulation. In a recent project CHAM's PHOENICS solver was used to predict the initiation of scouring due to the influence of an artificial reef, for three different wave flow directions, 0^o, 20^o and 40^o.

Calculation of threshold shear stress for scouring to occur

Bed Roughness: The predicted flow field and the local shear stresses can be strongly influenced by the type of seabed material, and its surface roughness height, which is an input for the CFD simulation. Sediment size and type, as well as the bed grain, will influence the transport rate and the induced stresses at the surface. For this project the seabed material was taken as **'silt clay',** with a bed roughness height of 0.005mm, which was based on published data [1]. The grain size was calculated from bed roughness height using correlations presented in [2].

Threshold of motion: The scour process involves a threshold of motion which is defined as the critical bed shear stress, or critical velocity beyond which significant amount of grains begin to move. The regions where sand grains will be more susceptible to movement can be identified by calculating the threshold of shear stress,

Critical shear stress correlations for sediment transport given by Guo [3], based on the Shields-Rouse equation were used in this investigation.

Dimensionless grain size

$$d_* = \left[\frac{g\left(\frac{\rho_s}{\rho}-1\right)}{\nu^2}\right]^{\frac{1}{3}} d_{50}$$

Threshold shields parameter

$$\theta_{cr} = \frac{0.23}{d_*} + 0.054\left[1-\exp\left(-\frac{d_*^{0.85}}{23}\right)\right]$$

Threshold of motion

$$\tau_{cr} = \theta_{cr} * g\left(\rho_s-\rho\right)d_{50} \rightarrow \tau_{cr} = 0.15\ \mathrm{Pa}$$

Where, τ_{cr} = critical shear stress; ρ_s = density of sediment (2650kg/m^3); ρ = density of water (1000kg/m^3); g = acceleration due to gravity (9.81m/s^2); d_{50} = grain size; d_* = dimensionless grain size.

The calculated threshold shear stress value (τ_{cr} =0.15Pa) was compared with the predicted shear stress at the seabed surface. Any location with a shear stress higher than the threshold value is more likely to undergo scouring. In the event of scouring, the region with higher wall shear stress is more likely to form a pit, whereas a nearby region with lower shear stress might undergo accumulation.

CFD model: The artificial reef consisted of about 200 arrays of thin angled plates hanging from the top of a steel frame attached to a concrete slab which was submerged in the seabed. Each array contained about 20 angled plates vertically spaced 30mm apart. It was not practical to model all of these plates in detail. Therefore each stack was represented by a simplified porous blockage to replicate the resistance to flow and turbulence generation. Coefficients of pressure drop and turbulence for the porous blockage were obtained from smaller sub-models, and then the values calibrated to achieve the same pressure drop and turbulent properties for detailed and porous geometries. An additional investigation was performed using a hybrid model to understand if the reef modelled as a porous object could produce similar pressure drop and shear stresses near the sea bed. The hybrid model consisted of a single row of detailed angled plates covering the bottom 10% of the array (i.e. closest to the sea bed), with the remainder represented as porous blockages. Two simulations were performed at 0^o flow angle, one using just porous objects and the other with the hybrid model. The results for fully porous and hybrid models compared well, giving very similar pressure drop, turbulent viscosity and shear stresses near the seabed, confirming the validity of the porous approach.

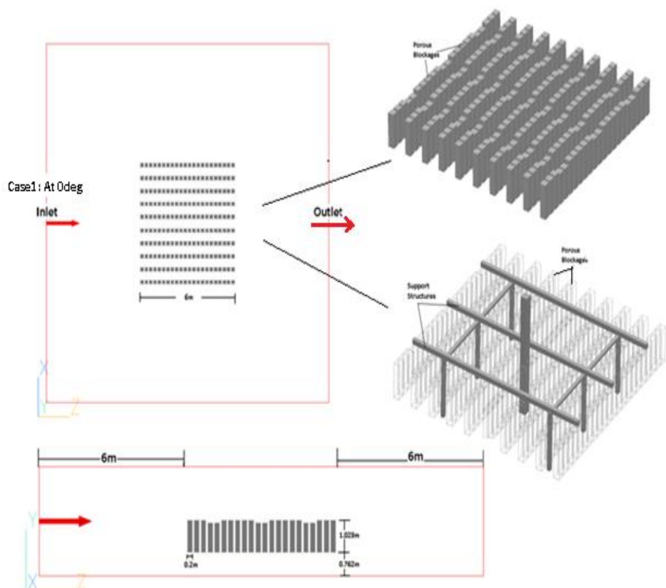


Figure 1 - CFD domain dimensions and zoomed in view of reefs (replaced with porous blockages) with and without support structures.

Setup: Each array of plates was replaced with a porous blockage, within a computational domain (18m x 18m x 3.5m) around the reef (6m x 6m x 2m high as shown in figure 1). The computation grid consisted of 0.8 million cells, and simulations were performed for three different flow directions i.e. 0^{o}, 20^{o} and 40^{o}. In the CFD model, the water velocity at the inlet varied with height using a logarithmic relationship, setting the reference velocity to 0.33m/s at a height of 3.5m, using an effective roughness height of 0.005mm (based on seabed material – silt clay). The surface of the seabed was assumed to be flat, and modelled as fully-rough with the same effective roughness height. Simulations were performed, with and without the support structures in place (as shown in figure 1) to assess their effect.

Results: For 0^{o} and 20^{o} flow angles, the predicted shear stress near the seabed was higher than the critical shear stress near the support structures (RST values >1 see Figure 2). The maximum shear stress was adjacent to the central column for 0^{o} flow, and then more evenly distributed for 20^{o}. For the 40^{o} flow, some scouring was predicted without the support structure, due to the angled plates alone; with the support structure, no scouring was evident for the upstream triangular region, but in the downstream region the most scouring of any case was predicted.

Estimation of scour depth: The aim of this project was to predict the initiation of scouring, rather than the actual transportation of particles. However, the depth of the scour hole was estimated using an empirical relationship ($S_c=2*D$, where D is the pile diameter), which is based on a single pile in open water.

Predicted contours of shear stress showed maximum scouring behind the support structures. Therefore, the scour depth based on the centre column (0.2m x 0.2m) and other columns (0.1m x 0.1m) was estimated to be 0.4m and 0.2m respectively using the analytical approach. N.B. CFD analysis on a single centre support (0.2m x 0.2m) in open water indicated the maximum predicted shear stress was less than the critical shear stress, so no scouring should occur.

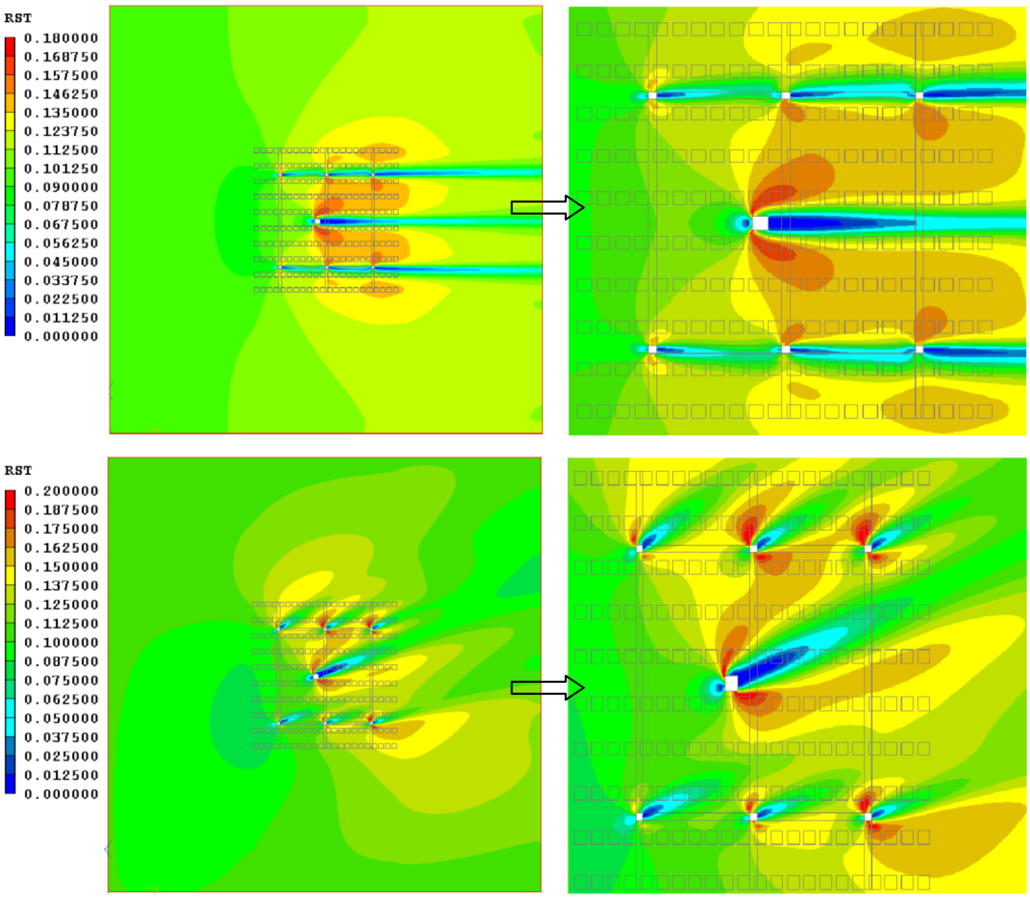
Conclusions: The study demonstrated the capability of PHOENICS to predict the initiation of scouring due to the presence of an artificial reef on the seabed by comparing predicted shear stresses at the seabed against critical/threshold levels above which scouring would occur. For all flow angles modelled, some local scouring was predicted to occur adjacent to the support structures. However, for the largest flow angle simulated (40^{o}), scouring was predicted across a wider region (even without the support structures) with shear stress levels near the seabed greater than the critical shear stress.

References:

[1] DET NORSKE VERITAS AS, On-Bottom Stability Design of Submarine Pipelines, October 2010, DNV-RP-F109.

[2] Richard Soulsby , Dynamics of marine sands: a manual for practical applications, page 47-48

[3] Guo, J., Hunter Rouse and Shields Diagram, Advances in Hydraulic and Water Engineering, Vol. 2, 2002, pp. 1096-1098



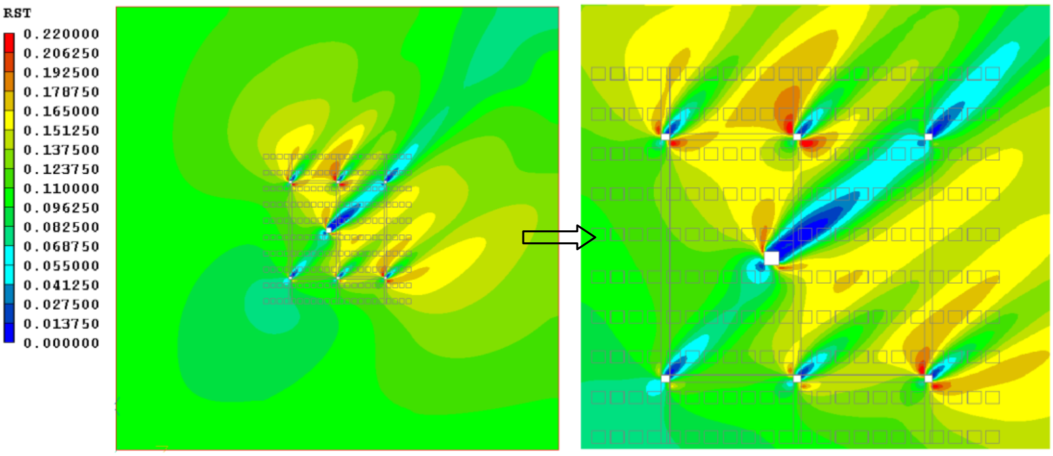


Figure 2 - Comparison of full geometry contours of shear stress taken near sea bed at three different angles of attack (0^0, 20^0 and 40^0).

3.2 Wind Energy Optimization

(Editor's note: CHAM has been a partner in a Eurostars Wind Energy project and the following article, relating to it, is extracted from the Newsletter of the lead partner on the project - WindSim AS)

Eurostars | Wind energy optimization

Together with our two partners Concentration Heat And Momentum Limited and Iberdrola Ingenieria y Construccion, WindSim AS has completed the project "Wind energy optimization by numerical wind modeling". The project was funded through the Eurostars program which is a European Joint Programme dedicated to the R&D performing SMEs.

The R&D challenges we attacked during the 18 months long project period spanned widely, including:

- Synthesizing simulation and remote sensing
- Wake modelling with actuator disc concept
- Robust and faster simulations
- Meso - micro coupling
- Validation

Development of improved numerical methods within micro-siting requires extensive testing, hence a significant part of the project was spent on validation. We gratefully acknowledge the contribution from Iberdrola and the other stakeholders, Mighty River Power, Nord-Trondelag Elektrisitetsverk, Statkraft and Statoil that delivered validation data

CHAM
PHOENICS- Your Gateway to Successful CFD

4) User Applications

4.1 Teaching with PHOENICS at MIT

by Professor Leslie K Norford

"Over the last 12 months, I have used the 2010 version of PHOENICS as an invaluable teaching tool in two courses: Architecture 4.411 Building technology Laboratory, and Architecture 4.423 Architectural Thermal and Fluid Dynamics.

The Building Technology Laboratory satisfies the Institute laboratory requirement for undergraduate students. The enrolment typically favours students in my department, Architecture, but attracts a few students from engineering or science disciplines. The curriculum consists of a set of 3 projects that have long focused on buildings in developing countries and very recently have further focused on schools. For two years the projects supported the work of an NGO working to construct low-cost schools in Sierra Leone. In the last year, the emphasis shifted to Haiti, where we provided Architecture of Humanity with some analysis of airflow and natural lighting in schools it is designing and building in the aftermath of the 2010 earthquake.

The schools in question will not have electricity, at least initially. Adequate light for the students to perform their assigned tasks and ventilation to provide conditions as comfortable as possible must therefore be achieved via the architecture and not mechanical or electrical systems. The MIT students explored ventilation through a combination of wind-tunnel tests of reduced-scale models and simulations with nodal airflow models and with computational fluid dynamics. The CFD studies were made with PHOENICS and provided detail about airflow patterns within the classrooms that we could not otherwise observe or simulate. With heat sources introduced in the simulations to represent the occupants, the students could also estimate indoor-outdoor temperature differences and determine the magnitudes of both wind-driven and buoyancy-driven airflows. I have posted the (unpublished) final reports and the tutorial document I provide to introduce the students to PHOENICS.

I use a similar tutorial in the Architectural Thermal and Fluid Dynamics course. This is a graduate-level subject that attracts engineering and architecture students, although the latter are primarily in our Building Technology program and have strong background in engineering or physical sciences. We approach airflow through analyses based on fluid mechanics, nodal simulations and CFD. For the latter, we develop the Navier-Stokes equations and pay attention to various representations of the Reynolds stresses, particularly zero-equation and k-epsilon methods. Students are given a homework assignment with PHOENICS and many use the program in their projects. Last year's project centered on renovation of a space on the MIT or Harvard campus to promote natural ventilation.

We have tentative plans to use PHOENICS at urban scale in the future and very much appreciate your continued support of our academic endeavours."

Professor of Building Technology, Associate Head, Department of Architecture, MIT School of Architecture & Planning
Tel (617) 253 1876

4.2 Teaching with PHOENICS at King's College London

by Professor Younis

(Editor's Note: The following is extracted from Professor Younis' letter to CHAM and is followed by the abstract of a paper, based on the course, prepared by a Student from the Engineering Division.)

"Our term at King's has come to an end and the students on the CFD course have now submitted the second (and final) project report detailing the use of PHOENICS to calculate the turbulent flow in a plane asymmetric diffuser.

A sample report is attached. Looking at some of the project reports, it is hard to believe that the students who wrote them had not even heard of CFD until late in January, or have had any meaningful previous experience with computational software of any kind. That many have submitted reports of a standard that a few years ago would have been publishable in an international conference is testimony of the ease of use and built-in capabilities of PHOENICS, and of the excellent hands-on support that was so skilfully and patiently provided by Rama.

Thank you all for making this course possible. This course will be offered again at King's in 2013 and I feel sure that my successor will want to renew the existing arrangements with CHAM

Thank you again for all your help, and best wishes."

Professor, Division of Engineering, King's College London
K1190418@kcl.ac.uk

4.2.1 Computational Analysis of Flow Around an Asymmetric Diffuser

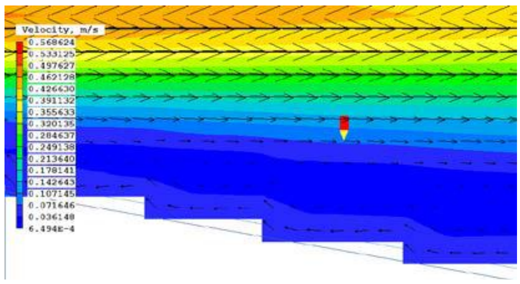
by Oyinkasansola Adeniji, MEng
Engineering Division, King's College London

Abstract

This study reports on the numerical investigation of the fluid dynamic flow in an asymmetric two-dimensional diffuser using the CFD code PHOENICS. The study will focus on turbulent flow in two-dimensions with smooth walls, also neglecting effects of temperature and compressibility. The first part of the study focuses on calibrating the CFD code and test conditions, and then moves onto analysis of factors which affect the separation and reattachment points. These factors include the angle of the diffuser, Reynolds number and different turbulence models. The study presents details of the computations performed, and reports on tests carried out to check the accuracy of the results. Comparisons are made between the present results and those from the experimental results of Buice and Eaton (1997).

(Editor's Note: The following figures from the report have been chosen arbitrarily to illustrate content.)

a)



b)

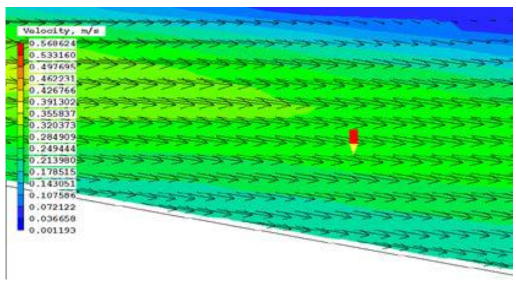


Figure 4 - Images of velocity flow contours with flow direction arrows with a) PARSOL off and b) PARSOL on

4.3 PHOENICS Use at Morelia Technological University from 2008 - 2012

by Alberto N. Conejo, Professor

1. Introduction

For the first time, I am submitting a technical report on the use of PHOENICS. The period of this report covers from 2008 until May 2012. I am currently in charge of the license in our Institute. Our Institute is part of a National System of Technological Institutes, coordinated by a central organism, the National System of Technological Higher Education (SNEST).

Our research group is part of the graduate program in metallurgy. We provide education at the masters of science level.

2. Report of activities

During this period, two major projects have been funded by the Mexican Government, through the National Council for Science and Technology (CONACYT). Seven students have obtained their masters degrees and two are currently enrolled. I am the thesis supervisor and as main co-supervisor is Prof. Marco Ramirez.

Research project 1:
Mathematical modeling of DRI melting in the Electric Arc Furnace
Funding: 63,000 euros
Sponsor: Federal government
Period: 2007-2010

Students who completed their master's program:

- O.J.P. Gonzalez,
- Y.I.C. Guzman
- J.L.G. Sanchez
- M. S. C. Lopez
- E. Nava (final thesis dissertation pending)

Research project 2:
Physical and mathematical modeling of bottom gas injection in metallurgical ladles with a top layer
Funding: 73,000 euros
Sponsor: Federal government
Period: 2011-2014

Students who completed their master's program:

- F.D. Maldonado-Parra
- D. A. N. Altamirano
- Amaro (PhD thesis to be completed in 2013)
- J.G. Gomez (initiating thesis work, to be completed in 2014)

3. Publications

The following publications have been included in international journals:

(Editor's Note: PHOENICS has been used for work described in the publications. Abstracts are included where the papers have been provided to CHAM)

MELTING BEHAVIOR OF SIMULATED DRI IN LIQUID STEEL
O.J.P.Gonzalez, Y.I.C. Guzman, M.A. Ramírez-Argaez and **A.N. Conejo.** Archives of metallurgy and materials, Vol 53, Nr 2, 2008, 359-364, Editor: Polish Academy of Sciences. Poland. ISSN 1733-3490.

Abstract: The melting process of sponge iron in electric arc furnaces involves highly complex phenomena of fluid flow, heat and mass transfer in unsteady state conditions. Few attempts worldwide have been carried out to couple these phenomena to describe the melting kinetics. In this work, some preliminary results are presented coupling three models: an arc model for AC electric arcs, a bath model and a melting model. The results describe the influence of some process variables on the melting kinetics of solid particles. The formation of the solid shell around metallic particles is expressed as a function of the initial particle size and arc length.

POWER DELIVERY FROM THE ARC IN AC ELECTRIC ARC FURNACES WITH DIFFERENT GAS ATMOSPHERES.
J.L.G. Sanchez, M.A. Ramírez-Argaez and **A.N. Conejo.** Steel Research International, Vol 80, Nr 2, January 2009, 113-120, Germany

Abstract: The amount of heat supplied to the electric arc furnace affects the melting rate. Power delivery in a three-phase EAF is influenced not only by the electric parameters such as arc length and voltage but also by thermal properties of the gases that form the plasma in the arc region. Application of the Channel Arc Model (CAM) suggests that power delivery is enhanced with long-arc, maximum tap operation and plasma gases with high heat capacity. It is also suggested that foaming improves power delivery due to the presence of gases with high heat capacity.

EFFECT OF ARC LENGTH ON FLUID FLOW AND MIXING PHENOMENA IN AC ELECTRIC ARC FURNACES.
O.J.P.Gonzalez, M.A. Ramírez-Argaez and **A.N. Conejo.** ISIJ International, Vol 50 (2010) Nr. 1, pp. 1-8, Japan.

MATHEMATICAL MODELING OF THE MELTING RATE OF METALLIC PARTICLES IN THE ELECTRIC ARC FURNACE.
O.J.P.Gonzalez, M.A. Ramírez-Argaez and **A.N. Conejo.** ISIJ International, Vol 50 (2010) Nr. 1, pp. 9-16, Japan.

Abstract: A computational fluid dynamics model coupled to a lagrangian model of melting/solidifying particles has been developed to describe the melting kinetics of metallic particles in an industrial Electric Arc Furnace (EAF), assuming that liquid steel occupies the entire computational domain. The metallic particles represent Direct Reduced Iron (DRI). The use of two previous models, an arc model and a fluid flow model, has made it possible to evaluate the melting rate of injected DRI in a three-phase EAF, evaluating the influence of the initial particle size, the initial DRI temperature, feeding position, feeding rate, arc length and some of the metallurgical properties of DRI. The frozen shell formed in the early stages of the melting process has also been evaluated in this model.

INFLUENCE OF THE TOP SLAG LAYER ON THE FLOWDYNAMICS IN AC-ELECTRIC ARC FURNACES.
M.A. Ramírez-Argaez, **A.N. Conejo**, Y. I. Guzmán and G. Trapaga. Int. J. Engineering Systems Modelling and simulation, Vol 50 (2010) Nr. 1, pp. 9-16, Japan.

Abstract: A mathematical model describing fluid flow for an industrial three-phase electric arc furnace has been developed which involves two immiscible liquids in the computational domain and the main driving force for the movement is due to buoyancy forces. Comparison between the case where only liquid steel occupies the entire computational domain and the case of two phases has also been carried out. The influence of slag physical and thermal properties on fluid flow and heat transfer is included in this work. The results of this work provide a clear description of flow patterns in the presence of the top slag layer. It has been found a marked effect of the slag phase to decrease the velocity patterns of liquid steel.

EFFECT OF BOTH RADIAL POSITION AND NUMBER OF POROUS PLUGS ON CHEMICAL AND THERMAL MIXING IN AN INDUSTRIAL LADLE INVOLVING TWO PHASE FLOW
F.D. Maldonado-Parra. M.A. Ramirez-Argaez, A.N. Conejo and C. Gonzalez. ISIJ International, Vol 51 (2011) Nr. 7, pp. 1110-1118, Japan.

Abstract: Gas injection in metallurgical vessels is an important tool to improve chemical and thermal mixing. Chemical mixing has been extensively studied in the past 40 years; however thermal mixing is still poorly understood. This work reports a mathematical model developed to describe the effect of the number and position of porous plugs on thermal and chemical mixing under industrial conditions. A relevant contribution of this work is the evidence indicating a suppressing effect of bottom gas injection on thermal homogenization with off-center gas injection; furthermore, it also suggests that mixing time is optimized with only one nozzle instead of two or three.

EFFECT OF FOAMY SLAG HEIGHT ON HOT SPOTS FORMATION INSIDE THE ELECTRIC ARC FURNACE BASED ON A RADIATION MODEL
J. L. G. Sanchez, A. N. Conejo and M. A. Ramirez-Argaez. ISIJ International, No.5,Vol.52 (2012).

Abstract: Recent Electric Arc Furnaces are equipped with ultra high power transformers to provide maximum values of electric power and minimize the melting time. The active power is increased by increasing arc length and arc voltage; however in these conditions energy losses due to radiation can also be increased with a consequent decrease in thermal efficiency. The energy radiated from the electric arcs is transferred to the walls inside the EAF promoting hot spots which represent a catastrophic operational condition. This work reports a radiation model which describes the formation of hot spots as a function of arc length and foamy slag height in an industrial EAF of 210 ton of nominal capacity. Temperature profiles on the surface of the water-cooled panels and values for the incident radiation were computed as a function of foamy slag height, used subsequently to define conditions to eliminate the formation of hot spots.

Alberto N. Conejo, Professor
Graduate program in metallurgy
58120 Morelia, México
anconejo@gmail.com

4.4 Research Work Using PHOENICS at NREC

by Steven Beale, NREC

Research work with PHOENICS this past year has been mostly concerned with ongoing activities on mass transfer in osmotic membranes, and also transport in solid oxide fuel cells.

The former relate to the use of periodic boundary conditions to obtain solutions to low and high-rate mass transfer in osmotic membranes under arbitrary (neither constant flux nor constant value) boundary conditions.

The latter involves porting earlier PHOENICS code based in GROUND-implemented FORTRAN to the more modern InForm meta-language thereby facilitating parallel processing for large-scale simulations.

Reference 1 is for the special edition of JHT in honour of Spalding's Franklin medal, reference 2 does not involve extensive use of PHOENICS, but rather was based on some discussions between the author, and Patankar and Spalding on turbulent flow in heat exchangers.

1) Numerical Study of Laminar Flow and Mass Transfer for In-line Spacer-filled Passages. S B Beale, J G Pharoah, A Kumar, ASME Journal of Heat Transfer, in preparation, 2012.

2) A Simple, Effective, Viscosity Formulation for Turbulent Flow and Heat Transfer in Compact Heat Exchangers, S B Beale, Heat Transfer Engineering, volume 33, issue 1, 2012.

Steven Beale Ph.D P.Eng F.IMechE F.ASME
National Research Council, Canada
Tel: (613) 993-3487, Email: steven.beale@nrc-cnrc.gc.ca

4.5 Concentration Distribution and Pressure Gradient of Particle-Water Slurry Flows in Horizontal Pipes

by Gianandrea Vittorio Messa, IIAR, Polytechnic School of Milan, Italy

(Editor's Note: As part of a collaboration between CHAM and Milan Polytechnic, one of Professor Malavesi's research students visited CHAM. This article is based on his work with PHOENICS.)

"Introduction

During a second period of stay at CHAM, in the context of my PhD in Hydraulics as part of the research group of Prof. Stefano Malavasi, I continued working on the numerical modeling of slurry flow of water and solid particles in horizontal pipes. Such flows are commonly encountered in many engineering applications, such as chemical, oil, mining and construction industries. Pressure gradient and concentration distribution have been the major concern of researchers, because they dictate the selection of pump capacity, and may be used to determine parameters of direct importance such as mixture and solid flow rates as well as secondary effects like wall abrasion and particle degradation.

The high economic and technical burden of experimental tests and the limitations of the existing simplified analytical and one-dimensional models have made CFD an attractive alternative in recent years. However, the CFD models considered in previous studies either appear inaccurate under certain flow conditions [1] or require very long simulation time [2], which makes them unsuitable for industrial applications. Therefore, the development of a suitable numerical model is the subject of my PhD research and the objective of my work whilst at CHAM.

Starting from a literature search, I customized the two-fluid IPSA model available in PHOENICS, adding specific constitutive equations and boundary conditions. The results – in terms of both concentration distribution and pressure gradient – were compared to experimental data available in literature [3,4,5], over a wide range of operating conditions: average solids concentration between 10% and 40% by volume; uniform particle size between 90 and 520 μm; slurry velocities between 1 m/s and 5.5 m/s; and pipe diameters between 50 and 150 mm.

Numerical model and boundary conditions

The Two-Fluid model was obtained by adding the following two features to the original IPSA model so as to reproduce the flow correctly:

- Mixture Viscosity: A correlation for the viscosity of the mixture, was used to define the particle Reynolds number in the interphase drag law. Several expressions are available in literature, and that of Mooney [6] was fount to best fit the experimental data:

$$\mu_m = \rho_C \nu_{l,C} \exp\left(\frac{[\eta]\alpha_p}{1-\alpha_p/\alpha_{pm}}\right) \quad (1)$$

  where which $[\eta]$ is the intrinsic viscosity, taken equal to 2.5 as suggested for spherical particles; α_{pm} is the maximum packing concentration, taken as 0.7; ρ_C is the density of the carrier fluid phase; and $\nu_{l,C}$ is the laminar kinematic viscosity of the carrier-fluid phase.

- Drag force: The drag force law is related to the particle Reynolds number according to the Standard Drag Law correlation available in PHOENICS, but the particle Reynolds number is defined with respect to the viscosity of the mixture instead of that of the carrier fluid phase. Therefore, $\mathrm{Re}_p = \rho_C d_p |\mathbf{U}_r| / \mu_m$, where d_p and $\mathbf{U}_r$ are the particle diameter and the slip velocity vector respectively. This modification is necessary to describe the phenomenon whenever in some cells the solid volume fraction approaches the maximum packing value.

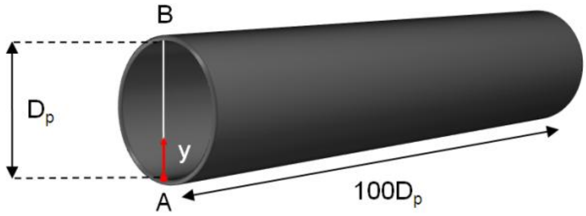


Figure 1 Sketch of the problem

Figure 1 shows the geometry of the problem; the computational domain covers only half of the pipe section due to the symmetry of the phenomenon. A fully-developed turbulent flow profile is applied at the inlet. No slip is assumed between the phases. The inlet volume fraction of the solids is taken as uniform. At the outlet, the normal gradient of all variables, and the value of the pressure are set to zero. The length of the computational domain, equal to 100 pipe diameters, is sufficient to ensure the attainment of fully-developed flow conditions.

At the pipe wall, no slip conditions are imposed to the carrier-fluid phase, and a logarithmic-law wall function is applied in the near wall cells. The proper wall boundary conditions for the solid phase are still a matter of discussion in literature. Two alternatives have been considered. At first, a zero-flux condition is applied to the particles. Afterwards, to account for particle-particle and particle-wall interactions, a Bagnold-type shear stress is imposed. In particular, the following term, derived from the model of Shook and Bartosik [7], is introduced in the momentum equation of the particle phase:

$$\tau_B = \left(\frac{8.3018}{\mathrm{Re}^{2.317}}\right)\rho_p d_p^2 \left[\left(\frac{\alpha_{pm}}{\alpha_p}\right)^{1/3} - 1\right]^{-1.5} \left(\frac{\tau_{w,C}}{\rho_C \nu_{l,C}}\right)^2 \quad (2)$$

in which: $\mathrm{Re} = D_p U_s / \nu_{l,C}$ is the bulk Reynolds number, defined with respect to pipe diameter D_p and slurry bulk-mean velocity U_s; ρ_p is the density of the particles; and $\tau_{w,C}$ is the wall shear stress of the liquid phase.

Results

When imposing a zero-flux wall-condition to the particle phase, the predictions of the two-fluid model show good agreement with the experimental evidence in terms of solid volume fraction distribution. As an example, Figure 2 reports results for the flow conditions considered by Gillies et al. [4], i.e. $D_p = 0.1027\ \mathrm{m}$, $\rho_p = 2650\ \mathrm{kg/m^3}$, $d_p = 270\ \mu\mathrm{m}$, $U_s = 2.6\ \mathrm{m/s}$ and mean solid volume fraction from 12% to 41%.

The contour plots of Figure 2 highlight the gradual accumulation of the particles as the mean solid volume fraction increases, phenomena that can be correctly reproduced by applying the above mentioned modifications to the original IPSA model. The solid volume fraction profiles along the vertical diameter (AB in Figure 1) appear in quantitative agreement with the experimental data of Gillies et al. [4], as shown in Figure 3.

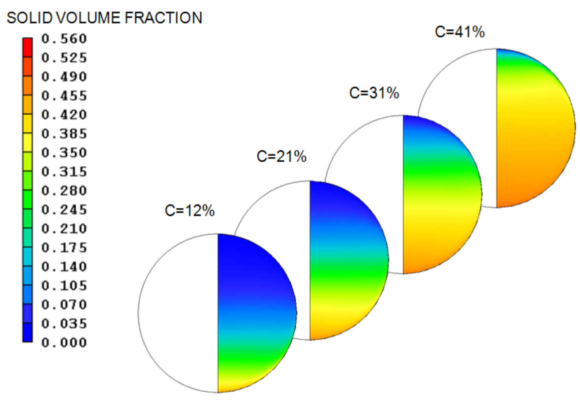


Figure 2 Contour plots for particle volume fraction

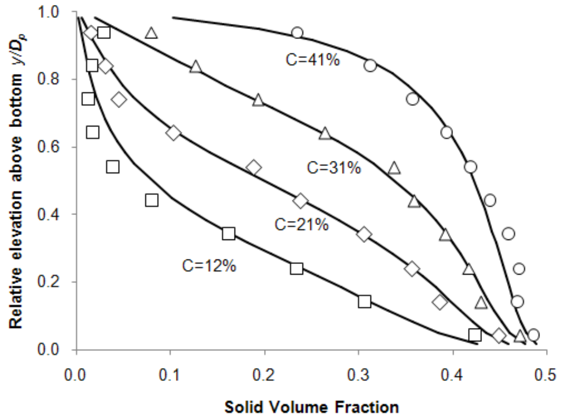


Figure 3 Particle concentration profiles: comparison between numerical predictions & experimental data of Gillies et al. [4].

The wall boundary condition of the solid phase is the key parameter affecting the friction losses. When imposing a zero-flux condition, the model does not reproduce the increase in pressure losses due to the presence of the particles, resulting in an underestimation of the pressure gradient which increases with the particle concentration, reaching up to about 50% for highly-concentrated slurries.

The introduction of the Bagnold stresses term (Eq. 2) in the momentum equation of the solid phase captures the dependence of the pressure losses upon the particle concentration, keeping the underestimation of the pressure gradient below 30% for all the flow conditions considered. However, such improvement is often offset by a worsening of the concentration profile. Moreover, neither of the two boundary conditions is capable of reproducing the increase in pressure loss occurring when the velocity is lower than the limit deposit velocity, and a moving bed of particle forms at the bottom of the pipe. For such cases, the phenomenon involves different physical mechanisms that are not accounted for in the present model.

Figure 4 reports the pressure gradient versus the slurry bulk-mean velocity for the case of $D_p = 0.1027$ m, $\rho_p = 2650$ kg/m^3, $d_p = 270$ μm and $C = 21\%$, comparing the experimental data of Gillies et al. [4] to the numerical predictions obtained using the two boundary conditions.

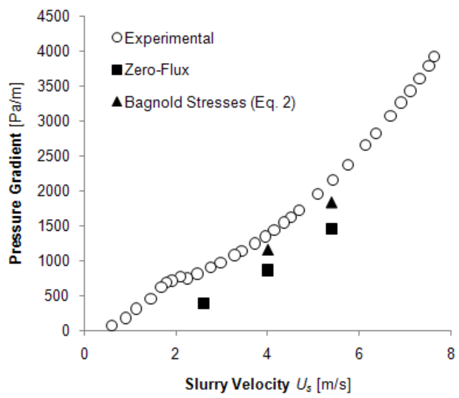


Figure 4. Pressure gradient versus slurry bulk-mean velocity: comparison between experimental data of Gillies et al. [4] & numerical predictions obtained using two boundary conditions

References

1) Lahiri, S.K., Gantha (2010). Slurry Flow Modelling by CFD. *Chem Ind. & Chem. Eng. Quaterly*, **16(4)**, 295-308.
2) Ekambara K., Sanders, R.S., Nandakumar, K., Masliyah, J.H. (2009). Hydrodynamic Simulation of Horizontal Slurry Pipeline Flow Using ANSYS-CFX. *Ind. Eng. Chem. Res.* **48(17)**, 8159-8171.
3) Roco, M.C., Shook, C.A. (1983). Modeling of Slurry Flow: The Effect of Particle Size. *Canadian J. Chem. Eng.* **61(4)**, 494-503.
4) Gillies, R.G., Shook, C.A., Xu, J. (2004). Modelling Heterogeneous Slurry Flow at High Velocities. *Canadian J. Chem. Eng.* **82(5)**, 1060-1065.
5) Kaushal, D.R., Tomita, Y. (2007). Experimental investigation for near-wall lift of coarser particles in slurry pipeline using γ-ray densitometer. *Powder Tech.* **172(3)**, 177-187.
6) Mooney, M.J. (1931). Explicit formulas for slip and fluidity. *J. Rheology* **2**, 210-222.
7) Shook, C.A., Bartosik, A.S. (1994). Particle-wall stresses in vertical slurry flows. *Powder Tech.* **81(2)**, 117-124."

4.6 Using INFORM to Model the Flow Created by a Paddle Propelled Carriage in Stratified Water

by Dr Robert Hornby

Introduction

It has been suggested that swimming may become more difficult when the sea or lake water is stratified because some of the swimming stroke energy goes into internal wave generation rather than propulsion of the body. Experiments conducted at the Royal Netherlands Institute for Sea Research have appeared to support this conjecture and numerical modelling has been used to advance understanding of the flow mechanisms (see PHOENICS Newsletter, Autumn 2010).

As part of this work, experiments have been conducted at the Royal Netherlands Institute for Sea Research using a paddle propelled carriage (of mass ~0.5kg) in a tank (length 2.04m, width 0.135m, water height 0.152m, see figure 1) containing first salt water (density 1025 kg/m^3) overlaid with fresh water (density 1000 kg/m^3) and then in the same tank with just fresh water. The paddles, (approximately 0.07m in length and 0.08m wide), which rotate at a fixed angular velocity (giving a period of about 2s), are meant to emulate (in a simple way) the action of a swimming stroke.

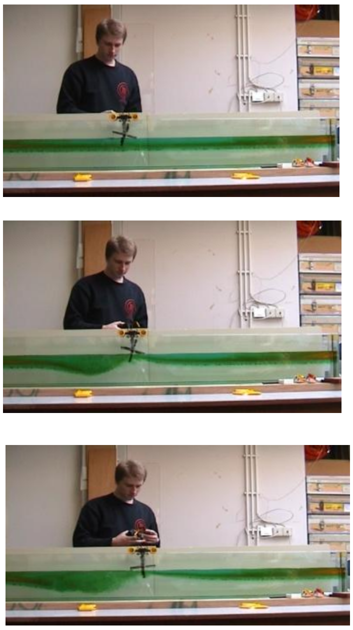


Figure 1 top to bottom (t=0s, 2s ,4s): development of the flow due to the paddle propelled carriage in a stratified tank (the top fresh water layer is dyed dark green). The carriage is moving from left to right.

The experiments show that there is a significant effect due to the stratification. In the stratified case the carriage velocity reduces (compared to the fresh water case) as an internal wave is created on the interface between the salt and fresh water (see middle figure of figure 1). It is also clear from the stratified experiment that significant mixing takes place downstream of the paddles as turbulent eddies are shed from the paddle edges.

These experiments were conducted principally as an illustration of the possible effect of stratification on forward swimming motion so there are insufficient details available to afford an accurate simulation. However, it was considered a worthwhile exercise to see if, with the limited amount of experimental information, PHOENICS could reproduce some of the experimental observations.

The flow field generated by the moving carriage was simulated using INFORM. The water surface in the tank has been treated as a rigid lid and initially the flow was assumed to be laminar. The paddles were simulated using four BOX objects (one for each paddle). Because the scale of the paddle-generated flow is small compared to tank dimensions, there is no economical fixed grid that allows the flow created by the moving carriage to be simulated accurately. Hence the carriage has been reduced to rest by applying an inflow equal but opposite to the current carriage velocity and a whole field body force derived from carriage acceleration. A fixed grid with refinement about the now stationary carriage was then used.

Deriving the fluid forces generated by the paddles

In order to calculate the carriage velocity and acceleration, the forces generated by the paddles in the direction of carriage motion must be known. The BOX object does not supply the force generated in the fluid so this has been calculated by applying a momentum balance (in the y direction) over a suitable control volume in the flow domain (figure 2). Similar balances can be used to find forces in other directions.

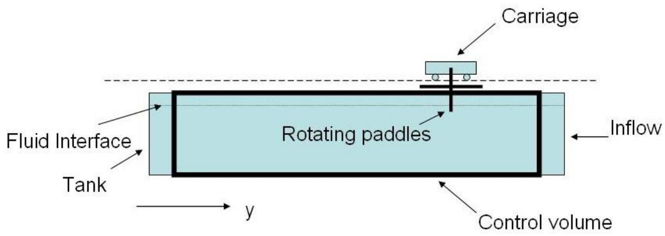


Figure 2. Control volume for momentum balance. An inflow equal to the carriage velocity has been applied on the North boundary tank face. On the opposite South tank wall a hydrostatic pressure outflow condition has been set.

In order to check the INFORM coding and the equivalence between the fixed and moving frames of reference two moving source cases in the tank with specified forces applied to the fluid (in the y direction) have been considered. For these simple cases a uniform grid in the x (lateral), y (axial) and z (vertical) direction is sufficient. In the first case a sinusoid source is used and in the second case an exponentially increasing source is used. Figure 3 shows that the INFORM coding applied to the momentum control volume correctly extracts the specified forces in the y direction. Also there is seen to be no significant difference between the extracted forces for the fixed and moving source cases, thus verifying the INFORM coding that reduces the carriage to rest. Comparisons of the carriage velocity, acceleration, distance travelled etc show similar agreement in each case with exact results derived from the specified source force.

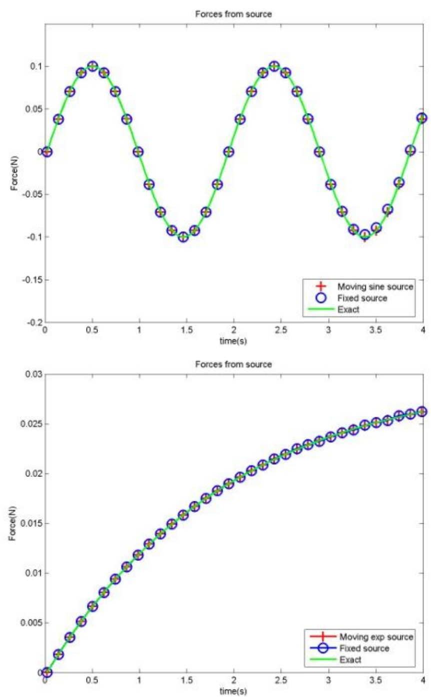


Figure 3. Top, sinusoid source. Bottom, exponential source. In each case simulations in stratified flow using INFORM have been conducted with the source moving and then with the source reduced to rest by applying the techniques described in the text.

A complete Q1 file for this simulation is available on request (see email address at the end of this text). Details of the INFORM language can be found in POLIS.

Results

The simulation has been run for eight seconds. Figure 4 shows a typical flow vector plot and the paddle positions determined by the array MARK.

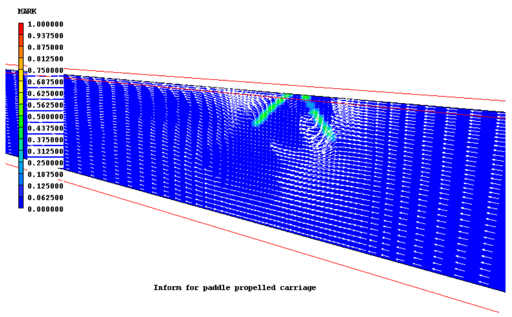


Figure 4. Snap shot of the flow showing paddles and velocity vectors.

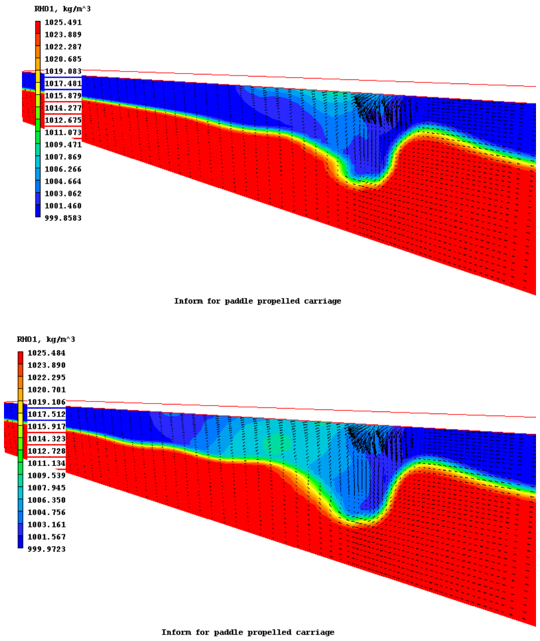


Figure 5. Density (red 1025kg/m^3, blue 1000kg/m^3) and flow vectors at t=2s (top) and t=4s (bottom) at tank wall.

Figure 5 shows the predicted density distribution after 2 and 4 seconds at the tank wall, so these plots can be compared with the experimental results shown in figure 1. In each case, an internal wave moving with the carriage is seen with very little upstream influence. The paddles produce a deepening of the wave trough and mixing of the fluid but not as much as is seen in the experiment.

Figure 6 shows the computed force on the carriage and the consequent carriage velocity. The force pattern is very regular (sometimes negative) and the velocity plot does not show the initial retardation of the carriage as mixing of the fluid column takes place. In fact the velocities achieved by the carriage resemble more the fresh water experiments than the stratified ones

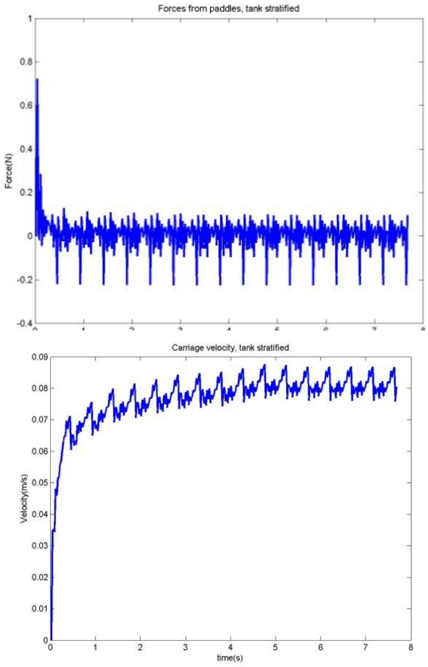


Figure 6. Top, force on carriage generated by paddles. Bottom, velocity of carriage.

It is clear from the stratified experiment that considerably more mixing takes place and this appears to be due to turbulent eddies shed from the paddle tips. It must be this additional (mixing) energy loss that accounts for the retardation of the carriage (since if it was just energy loss due to internal wave formation the laminar simulation should show it). In order to reproduce this effect, the simulation needs to model the effects of turbulence generated by the paddle motion and this requires a much finer grid and turbulence model or an empirical mixing model applied to the relatively coarse grid used here.

Dr R. P. Hornby e-mail: bob@hornby007.wanadoo.co.uk

If you use PHOENICS we would like to receive an article from you; please send contributions to Colleen King (cik@cham.co.uk).

Academic Users obtain PHOENICS under Special Terms and Conditions so we look forward to receiving:

1) ***An annual report regarding the CUSTOMER's use of PHOENICS under special discount terms on the anniversary of the licence either at the end of an annual licence period or at the time when annual maintenance is renewed.***
2) ***Q1 and GROUND files used in PHOENICS related teaching and/or research shall be provided periodically as they are generated and no less frequently than once per year with the annual report.***
3) ***Copies of all PHOENICS based publications as generated or no less frequently than once a year with the annual report.***

We look forward to hearing from you.