Autumn 2014

PHOENICS News

CHAM

PHOENICS – YOUR GATEWAY TO CFD SUCCESS

PHOENICS Goes Green in China

PHOENICS and Green Building Simulation

PHOENICS has, this year, been chosen as one of the CFD software codes being used to simulate environmental matters in Chongqing.

Engineers from Shanghai Feiyi trained Green Building Experts in Chongqing City at an official training course attended by 200 delegates from design institutes in the area.

Delegates at the Green Building Training Course in Chongqing City

Mr Yu, Shanghai Feiyi, training Green Building delegates

PHOENICS in HVAC and Green Building Evaluation

The 2014 HVAC thermal power academic meeting of the five north western provinces (Shanxi, Gansu, Qingai, Xinjiang Uygur and Ningxia Hui) was held at HanZhou Jiao Tong University from August 15 – 16. This is a most important regional meeting which was attended by academic leaders, design institutes, universities and business representatives. Shanghai Feiyi sponsored the meeting, participated in the exhibition and Mr Yu Zubin made a speech about the application of PHOENICS to green building evaluation.

Mr Yu, Shanghai Feiyi, at 2014 HVAC Thermal Power Academic Meeting Exhibition

Delegates learning about the application of PHOENICS to green building evaluation

For further information contact CHAM (sales@cham.co.uk) or CHAM's Agent in China, Shanghai Feiyi (www.shanghaifeiyi.cn).

PHOENICS Heatex

Alex Adam at CHAM

Heatex is a novel, adaptable plugin for PHOENICS that simulates the fluid dynamics and heat transfer properties of real world heat exchangers and is suited to both practical engineering and research.

Heatex solves for the flow and thermal properties of physically realistic, three dimensional heat exchangers. It is capable of simulating many existing industry standard designs (TEMA types) and allows inclusion of realistic complexities such as fouling, baffles and divider plates. Users are free to specify the dimensions of all constituent components, the tubeside/shellside fluids as well as boundary data as appropriate to their application.

Perhaps the greatest advantage of Heatex is the long term plan to eliminate the need to use dubious empirical constants in heat transfer equations. These will instead be computed iteratively from an initial guess by cyclically connecting large scale exchanger simulation to smaller scale modelling of the individual tubes.

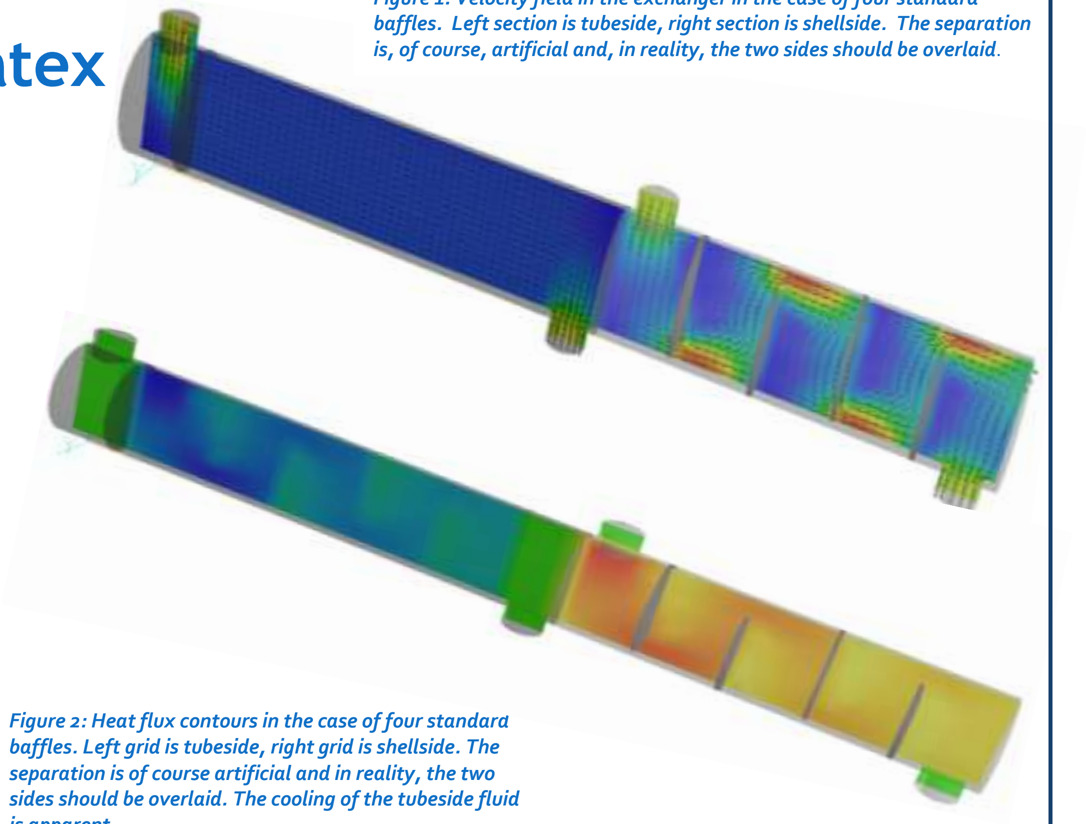


Figure 1: Velocity field in the exchanger in the case of four standard baffles. Left section is tubeside, right section is shellside. The separation is, of course, artificial and, in reality, the two sides should be overlaid.

Figure 2: Heat flux contours in the case of four standard baffles. Left grid is tubeside, right grid is shellside. The separation is of course artificial and in reality, the two sides should be overlaid. The cooling of the tubeside fluid is apparent.

FOR FURTHER INFORMATION CONTACT SALES@CHAM.CO.UK

Supply Chain Charter for Nuclear Decommissioning Sites

http://www.decommsupplyevent.co.uk/

Visit CHAM on Stand Number 152 at the NDA Estate Supply Chain Event to be held November 6 2014 at Event City, Manchester

PHOENICS goes Green in Brazil

Fabio Fundo, Safe Solutions, Brazil

Safe Solutions held a Course at Cesmac (Centro de Estudos Superiores de Maceió), to present PHOENICS and describe its use as an important tool for architecture and the urban environment.

Delegates were advised how the software can be used to optimize the use of natural resources in projects, as well as to simulate the possible gain in thermal comfort inside buildings.

Existing project cases were presented as well as theoretical examples showing air flow inside and outside structures.

The course was attended, and well received, by more than 30 students. Its success means that in-depth PHOENICS Courses may be scheduled for the coming months.

F1 in Schools Competition

Wilmington Grammar School for Boys

was victorious in the National Finals of the **F1 in Schools** competition.

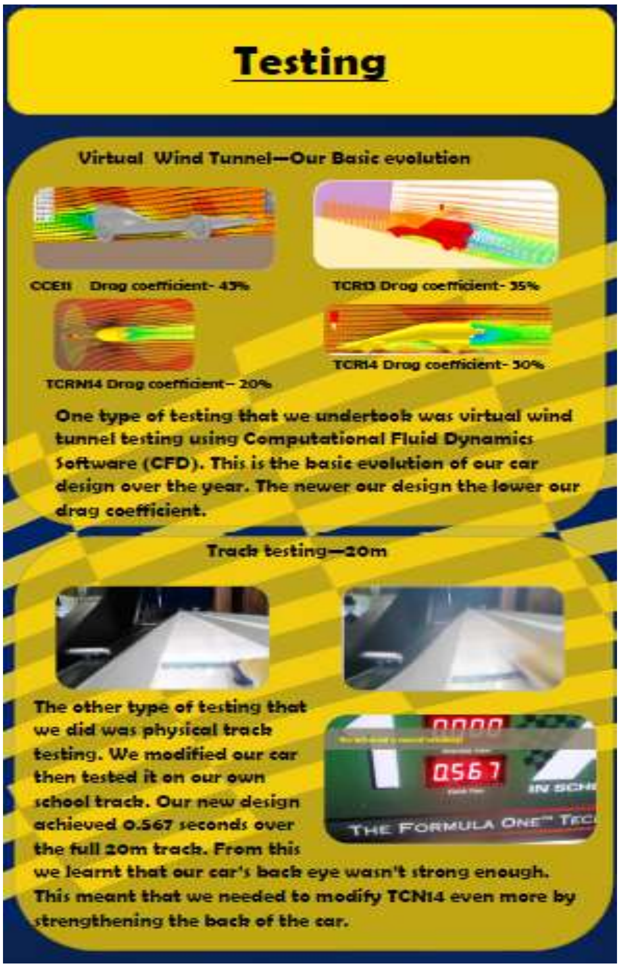
Its Bloodhound F1 team, **Turbocharged**, achieved success at the national Final held in Birmingham earlier this year.

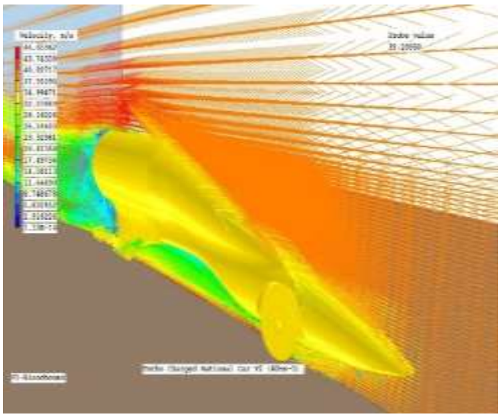
Wilmington Boys are amongst the more than 800 users of CHAM's F1-VWT package (a special-purpose sub-set of PHOENICS).

Team Turbocharged

Supervising teacher, Mr Neil Winter, reported that **Turbocharged** were awarded the trophy for Research and Development due to their F1-VWT-based car body development. They also won the trophy for Best Engineered Car and were crowned **National Champions**.

F1 in Schools Competition: www.f1inschools.co.uk



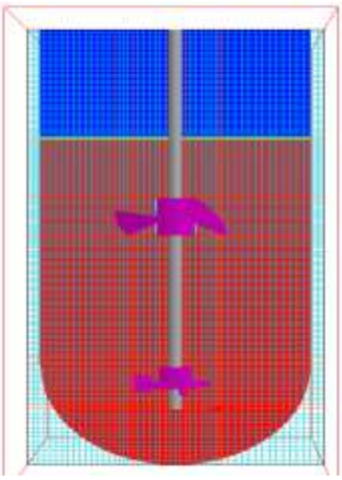


F1-VWT in Schools Is available from Denford Limited. For further information: http://website.denford.ltd.uk/.

For information on commercial, and non-school, use contact sales@cham.co.uk.

Optimization & Modelling of the Hydrodynamics of an Anaerobic Stirred Tank Reactor

Zaineb Trad and Christophe Vial, LABEX IMobs[1], Polytech Clermont-Ferrand-Institut Pascal, Axe GePEB, Bât. Polytech, 24 avenue de Landais, BP 20206, 63174 AUBIERE Cedex

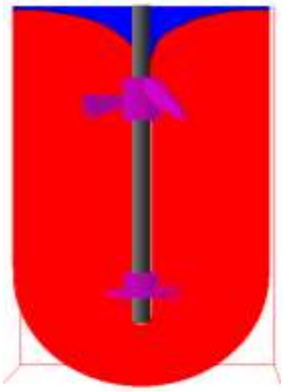


The project objective is to enhance and optimize the design of an anaerobic stirred tank bioreactor for scale-up purposes and to turn batch to continuous processing.

CFD, in this case, is a useful tool better to elucidate the complex interplay between reactor geometry and operating conditions on the one hand, and mixing and biochemical reaction on the other. The figures above illustrates the design of a 5-litre bioreactor when operated with suspended straw-like biomass (top), and of the 3D mesh (bottom: a structured grid with 58×58×120 cells) with the two turbine impellers devoted to liquid homogenization and solid suspension, respectively.

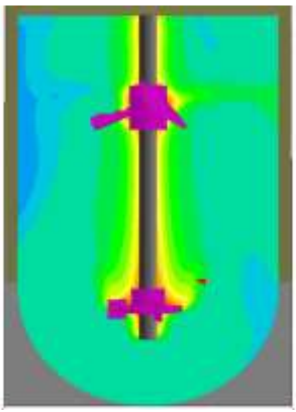
PHOENICS software was used, with MOFOR, to simulate the rotation of the impeller. The study was conducted in four steps:

1. MOFOR was coupled to SEM (Scalar Equation Model) to investigate vortex formation, as an un-baffled stirred tank is used to avoid stationary solids in the baffle region. Anaerobic fermentation is currently driven at low rotation speed, but a minimum value is required to maintain the suspension of solid biomass, as shown below. A compromise was found between mixing, solid suspension and vortex prevention.



Vortex Formation

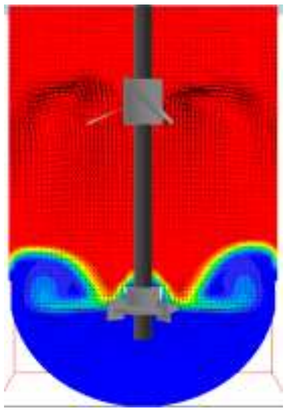
2. Mixing a passive tracer in the liquid phase was studied. Hydrodynamics were modelled using the k-ε Chen model for turbulence and the MUSCL discretization scheme which was shown to provide the best compromise between computation time and accuracy, even at low rotation speed. A typical tracer dispersion example is shown below. The objective was to improve the position of the impellers.



Concentration Profile

3. As biomass suspension occurred before vortex formation, this was simulated using the Algebraic Slip model.

Comparison with experimental data, as shown below, illustrates how the position of the bottom impeller induces solid suspension and enables the optimization of its position as a function of the amount of solid waste.



Onset of solid suspension

4. The last, and ongoing, step is to implement a simplified version of Anaerobic Digestion Model 1 (ADM1) (describing the biochemical conversion of biomass into bio-hydrogen, assuming that glucose and inorganic nitrogen are the substrates) into PHOENICS using In-Form. This constitutes a challenge, as coupling ADM1 with CFD was reported last year for the first time.

In conclusion, the comparison between experiment and 3D simulation has shown the ability of PHOENICS to model the multi-scale phenomena involving gas-liquid separated phases. A solid dispersed phase and biochemical reaction has been assessed but further work is needed to implement the complete structure of the ADM1 model.

[1] LABEX IMobS "Innovative Mobility: Smart and Sustainable Solutions" is a French Laboratory of Excellence dealing with efficient and environmentally-friendly technological building components for innovative mobility for the transport of persons, manufactured objects and machines. One of its challenges is to develop new innovative processes and technologies for the production of 2^{nd} and 3^{rd} generation biofuels. Particularly attractive for sustainability are 2^{nd} generation biofuels from food and agro-waste.

Special thanks go to Dr J. Ouazzani of Arcofluid for his support in the definition and setting up of the problem in the PHOENICS code.

CFD Analysis using PHOENICS of heat distribution in a high density passenger pick-up and drop-off point in Singapore

Venugopalan Raghavan of ZEB

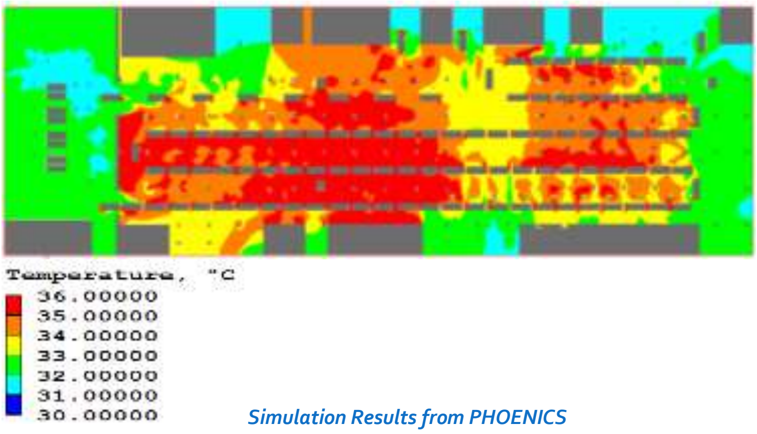
ZEB-Technology Singapore has been a user and a distributor of PHOENICS since 2009. A recent project involved a major upgrade of the mechanical ventilation system for a region with high circulation of passenger pick-ups and drop-offs. The HVAC consultants suggested the use of jet fans over the existing conventional supply and exhaust grille system.

ZEB was tasked with performing the CFD simulations that were requested by the client in order to have an idea of how the systems would perform and to compare the simulated results with measured values from the site.

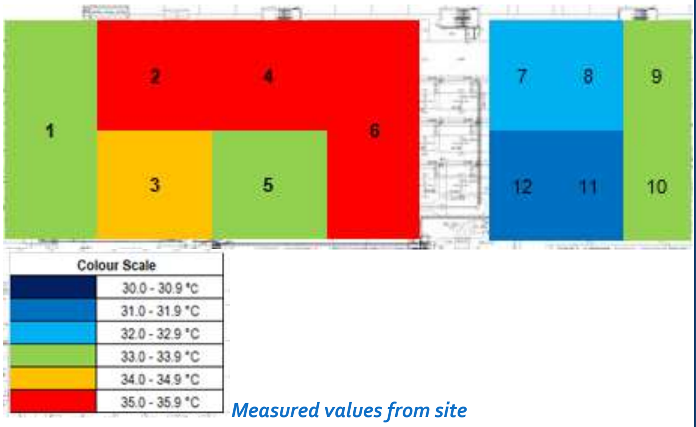
Simulations were done using PHOENICS, with the geometry modelled using built-in objects in PHOENICS.

Since only the heat and pollutant release from the vehicles were the focus areas, the vehicles themselves were modelled as blockages. The exhaust was modelled as an INLET object with the temperature, speed and concentration of pollutant being specified. The pollutants were specified using the extra variables option, which allows for creation and use of extra variables, especially those involved in passive scalar transport. Heat sources for the vehicles were modelled using the PEOPLE object which allows for a convenient way to model heat sources. About 110 vehicles were modelled for this exercise which was meant to simulate the worst case scenario with almost all of them being in the idling mode.

The results for the existing system were compared with measured values and the results showed a reasonable degree of match.



Simulation Results from PHOENICS



Measured values from site

PHOENICS & Rhino: A Natural Ventilation PlugIn

Bing Wang, School of Design, Harvard University

1. Introduction

Buildings account for 40 percent of energy use in the USA, Europe and Asia. The high level of energy consumption by buildings is a negative contributor to climate change and environment issues, placing pressure on designers to improve building performance. Natural ventilation is considered a viable technique for replacing or reducing active cooling loads and fan power in residential and commercial buildings. Mounting evidence suggests that occupants in naturally ventilated buildings accept higher temperatures than those in air-conditioned spaces. This heightened tolerance further increases potential energy saving through natural ventilation.

After years of research and practice, natural ventilation principles in buildings are well known. However, it remains difficult for designers to be confident about designing an impeccably functioning natural ventilation system. Evaluating natural ventilation requires users to have much experience and knowledge to handle simulation tools including Computational Fluid Dynamics (CFD) software, airflow networks and energy simulation. As the result, application and use of natural ventilation are rare.

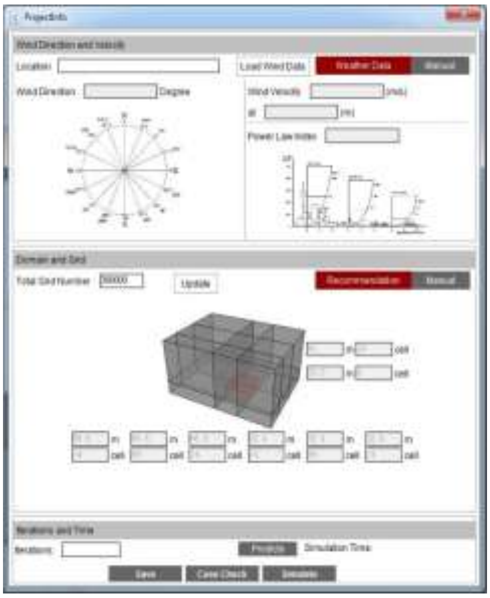
The importance of marrying design and evaluation tools during the schematic design phase is self-evident because decisions made at this point - building form for example - largely determine their natural ventilation potential. Simulation plugins in the modelling environment could help designers conduct building performance analysis and get visual feedback to optimize design. The Rhino plugin which is described here aims to help designers evaluate natural ventilation with CFD in the schematic design phase.

2. Plugin Interface

The plugin was developed as an interface of PHOENICS in Rhino (a widely used 3D modelling tool for designers), with step by step guide for designers to perform a natural ventilation evaluation. The plugin has three functions, Model, Case and Result, indicated by three menu buttons.

The Model function guides designers to prepare models for CFD simulation. To include a building in the CFD simulation, click on the Add button next to Building. Give a name to this building layer and click Select. The window will close and the user can choose a geometry in Rhino.

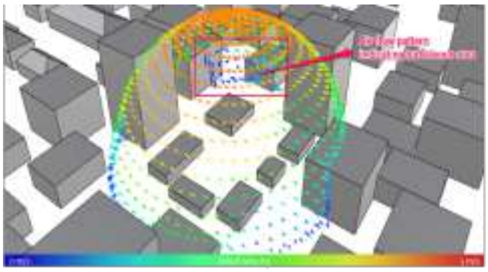
The Case function is the main part of the plugin. It simplifies inputs of a CFD simulation based on previous research conclusions. There are three parts in boundary condition settings – Wind Direction and Velocity, Domain and Grid, Iterations and Time. Designers could perform a CFD simulation just by filling all necessary information.



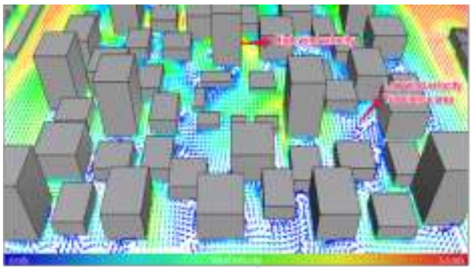
The Result function will visualize simulation results in Rhino, which will provide clear feedback to designers.

3. Plugin Result Examples

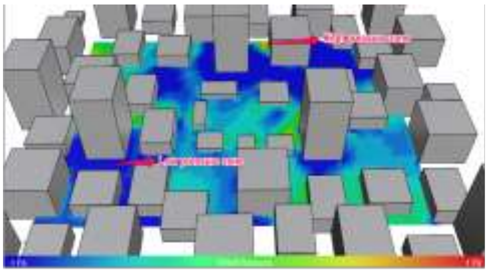
The following are example simulation results visualized in Rhino, including wind flow pattern, wind flow on pedestrian height, wind pressure on pedestrian height and wind pressure distribution on façades.



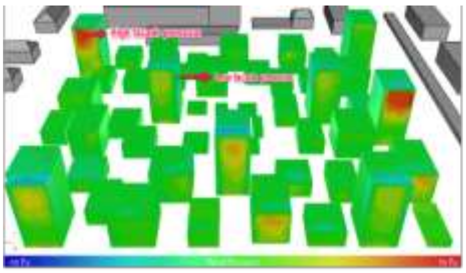
Wind flow pattern



Wind pattern at pedestrian height



Wind pressure on pedestrian height



Wind pressure on façades

4. Discussion

This plugin could help designers perform their own CFD simulations and obtain visualizations of natural ventilation. It provides information which enables designers to take natural ventilation into consideration in schematic design phase.

The plugin is not limited to natural ventilation. It provides easy access to detailed CFD simulation results which can support other outdoor environment analyses, including thermal comfort and air pollution. In future, it could be combined with tools, such as energy simulation, for further analysis in later design stages.

On the design side, the tool creates more possibilities. Current workflow could be introduced to Grasshopper for automatic design optimization.
Simulation technologies play an important role in the building industry. However, in most cases they are employed in the late stages of design where there is little room for change. Design has more flexibility in earlier stages and the importance of introducing simulation technologies in the early design stage is self-evident.

With the development of more simulation tools like this one, more information about building performance and built environment quality can be made available to designers at an early stage, enabling better informed decisions to be made at the schematic design phase.

Furthermore, designers could use these devices as educational tools to understand building physics and can even be inspired by the simulation results. This could help simulation become a new design methodology in the future.

To contribute to the PHOENICS Newsletter please send articles to news@cham.co.uk in Word format.

We look forward to receiving annual reports from Academic Users in the same format.

Please note that transmission of articles will be taken, by CHAM, as authorization to publish in the PHOENICS Newsletter.

Thank you.

Numerical Modelling of Pressure Drops in Micro-Channels with Squared and Staggered Pillar Arrangements

Jalil Ouazzani – ArcoFluid (Bordeaux, France), Yves Garrabos, Romain Guillaument, Carole Lecoutre, Samuel Marre & Sandy Morais – ICMCB (Pessac, France).

High-pressure micro-models are useful tools to visualize pore-scale fluid distribution and displacement by injecting, for example, CO_2 into a water-saturated micro porous media under geological reservoir conditions. ICMCB researchers performed fluid flow experiments in a well-designed two-dimensional pore network inside high-pressure silicon/Pyrex micro-models.

Such micro-models have a low pertinence with respect to the local complexity encountered in 3D porous media of geological formations and measurements of local velocity field remain an unresolved problem, especially at high pressure and high temperature. The 2D characteristics of these models allows observation of the behavior of two-phase distribution at different operating conditions for the first time in real p,T conditions,. The combination of flow rate and pressure measurements, wettability change of the solid walls, and a video recording of the pore network, can be used to capture some key mechanisms at a typical pore scale of a few tens of micrometers.

Numerical modelling of the experiments was carried out using PHOENICS software from CHAM.

To validate the numerical model to be used at ICMCB, it was tested against published experiments of water injection in micro-models at various Reynolds number, and plots arrangements (Measurement of pressure drop and flow resistance in micro-channels with integrated micro-pillars: Microfluid Nanofluid (2013) 14:711–721 - DOI 10.1007/s10404-012-1089-1)

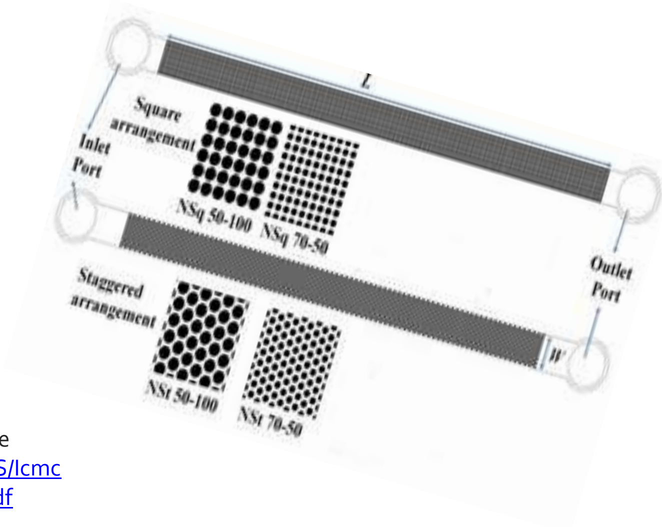
Initially the 2D models in PHOENICS were used but it was found that the pressure drop varied extensively from that obtained via experiment. This was mainly due to the short height of channel which meant that the friction effect from the top and bottom walls was substantial. Full 3D calculations were therefore used.

These are costly due the large amounts of pillars in the channel but reproduced the expected pressure drop satisfactorily.

Given the low height of rectangular channels in the study, the 2D-Helle-Shaw formulation was investigated and found to be in good agreement with the 3D model calculation results with a much lower CPU cost.

For a full copy of this paper see http://www.cham.co.uk/DOCS/Icmcb_Arcofluid_Autumn_2014.pdf

Fig 1: Micro-channel with integrated micro-pillars (MCIP) considered in the work. NSq refers to the square arrangement of pillars and NSt to the staggered one.



Arrangement	Pillar	Dimensions			Dimensions	
				Channels		
	d (µm) Diameter	S (µm) Distance between pillars		h (µm) height of channel	L (mm) Length of channel	W (µm) Width of channel
Nsq 70-50	50.2	82		98.4	24.46	808
Nsq 50-100	100	124		98.4	18.87	1250
Nst 70-50	50	87.2		98.4	24.26	869
Nst 50-100	100	135		98.4	20.11	1346

Table 1: Geometrical dimensions of different fabricated MCIPs considered in the present work.

CHAM

WHAT'S HAPPENING: PHOENICS Diary

When	What
On going	CHAM, London holds regular PHOENICS training courses at its Head Office in Wimbledon Village. Courses can also be arranged at User premises. Please see www.cham.co.uk or contact sales@cham.co.uk.
On going	Shanghai Feiyi, China, holds PHOENICS training courses in various cities across China as well as in Shanghai. Please see www.shanghaifeiyi.cn.
On going	C-h-a-m-p-i-o-n, Taiwan provides regular basic and advanced training. See www.cpet.com.tw or contact: sales@cpet.com.tw.
On going	Focus Advance Technologies, Malaysia provides training for beginners, intermediate and advanced CFD users. www.focus-technologies.com.my
On going	ACFDA, Canada, provides multi-level training to ensure that customers become knowledgeable users of CFD models and PHOENICS CFD software. Training sessions can be at the Toronto office, on client sites or over the internet. Contact info@acfda.org.
On going	ACS, USA, provides on-site training giving a detailed introduction to PHOENICS and assisting with tailored solutions. Training can also be provided off-site for companies or groups. Contact cfdconsu@cfd-consulting.org.
Oct 10	CHAM Japan PHOENICS User Meeting. www.phoenics.co.jp.
Oct 16-17	C-h-a-m-p-i-o-n, Taiwan Basic Training. Contact: sales@cpet.com.tw.
Nov 6	Concentration Heat & Momentum Limited, See CHAM on Stand 152 at the NDA Estate Supply Chain Event at Event City Manchester. For further information contact: sales@cham.co.uk.
Nov 13-14	C-h-a-m-p-i-o-n, Taiwan Advanced Course: HVAC. Contact: sales@cpet.com.tw.
Nov 25-27	PHOENICS/FLAIR Training Course, Singapore. For further information contact sales@cham.co.uk.

PHOENICS News Autumn Edition

Concentration, Heat & Momentum (CHAM) Limited
Bakery House
40 High Street
Wimbledon Village
London SW19 5AU
England
www.cham.co.uk
sales@cham.co.uk