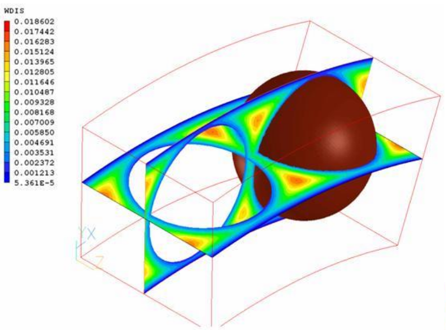
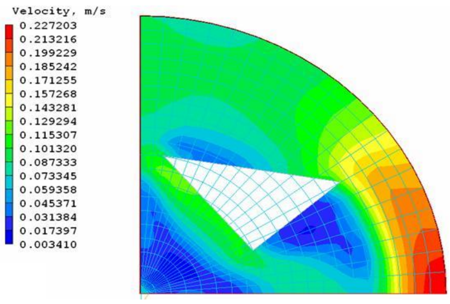
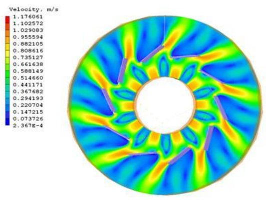
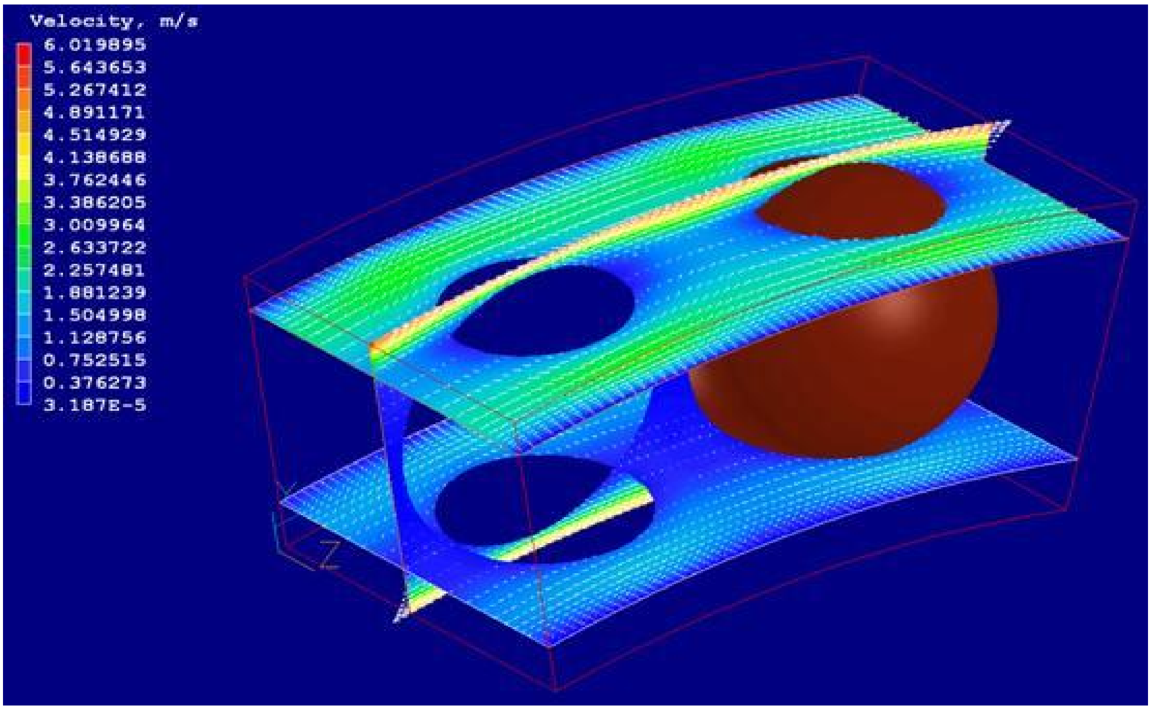
PHOENICS
News

Spring 2006









News from the pioneers of computational fluid dynamics

EDITORIAL

We're set for a new release

Over the past months I have been in contact with Agents to find out what Users want from PHOENICS so that I can feed this information back to the technical team here at Head Office.

Agents have stressed that you, the users, want to be able to use PARSOL more extensively than has been the case. We have worked on the use of PARSOL with Polar coordinates, as this facility is crucial for people interested in modelling rotary equipment or turbomachinery, for example.

To show what can be done Dr WeiShan Zhang at CHAM has created a simplified 3D turbine case with two rows of blades evenly distributed on a cylindrical axis and 26 blades in each row. There is an inlet with10 [m/s] axial velocity on the origin side and one thirteenth of the case is simulated with cyclic boundary conditions. The KeChen turbulence model is applied and the results are shown below.

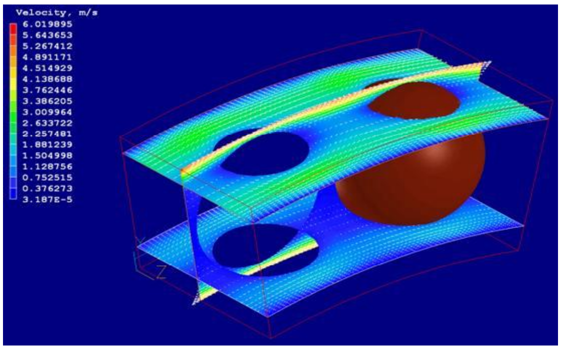
There are also visuals on the front page and pages 6 & 7 of the results for a 2D cyclic case with 10 inlets, velocity of 1.0 [m/s] in the radius direction where the inlets are evenly distributed on the inner circumferential face and outer circumferential boundary. One tenth of the case is simulated with cyclic boundary conditions and the LVEL turbulence model is used.

We will, in future communications, be introducing readers to our network of Agents worldwide. So that those of you who have not met each other can put faces to names that you hear, and which appear in the Newsletter, we are asking all Agents to supply us with photographs and a brief description of their company.

We would like to do the same thing with Users. When you provide with details of work you do with PHOENICS for publication in our Newsletter could you also supply photographs. These will be used to accompany the articles that are read with interest by other Users.

Colleen I King, Director and Company Secretary
cik@cham.co.uk

CONTENTS



REVIEW OF BUILDING TECHNOLOGY PROGRAMME AT MIT

MIT faculty and graduate students continue to use PHOENICS in their teaching and research. CFD instruction was provided in the spring term in a new course, Thermal and Fluid Dynamics in Buildings, taught by Prof. Leon Glicksman and Prof. Leslie Norford. Using PHOENICS, students worked on demonstration problems and tutorial cases in class, and later completed an airflow-analysis assignment with the program. In addition, several students used PHOENICS to support their final projects, which involved design and analysis of a ventilation system in a renovated suite of rooms on the MIT campus.

The research described below is by two, recently graduated, PhD students, Gang Tan and Christine Walker. Mr. Tan focused his research on naturally ventilated buildings by seeking integration of three typically distinct simulation methods: nodal airflow analysis, suitable for simulating flows between large numbers of zones within a building; thermal analysis, needed to provide the indoor temperatures required to compute buoyancy-driven flows; and CFD, needed to provide detailed spatial information about intra-room airflows. The integration of the first two methods is intended to provide architects and engineers with a tool that can be used quickly and easily in the early stages of a building's design. PHOENICS was used for a more detailed airflow analysis of spaces within the building. A key element was a series of comparisons of a full CFD simulation for a building and a combination of a nodal network and CFD simulation for one zone. Dr. Tan used a physical model of a building, constructed at 1:12 scale as the basis for his comparison. The building was a three-storey, naturally ventilated office building in Luton, UK.

A key question was how to model the atrium and the large openings that connect the atrium to the open-plan office areas, for buoyancy-driven flows. Two approaches are shown below; the left is a typical approach with single openings connecting the thermal zones and the right is a more detailed division of spaces. Based on comparison with PHOENICS simulations, the more detailed approach was found to be necessary to obtain accurate predictions of indoor temperatures; with this more detailed nodal simulation, temperature errors relative to the indoor-outdoor temperature difference were on the order of 10%.

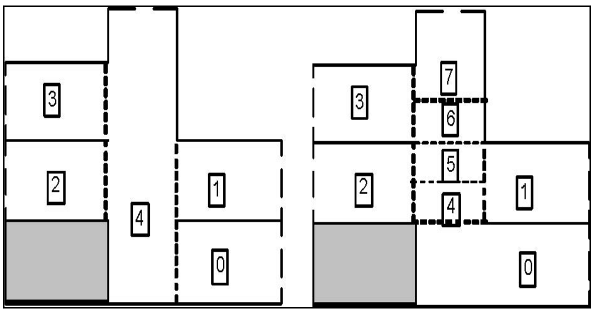


Fig 1. Zoning schemes for a nodal analysis of a naturally ventilated building.

Airflow patterns for the detailed nodal model were in good agreement with CFD simulations; results for buoyancy-driven flows are shown in Figure 2. Note that air flows into the building via the ground and first stories and leaves not only via the atrium but also from the second-story windows. The latter is not desirable, because occupants will breathe stale rather than fresh air.

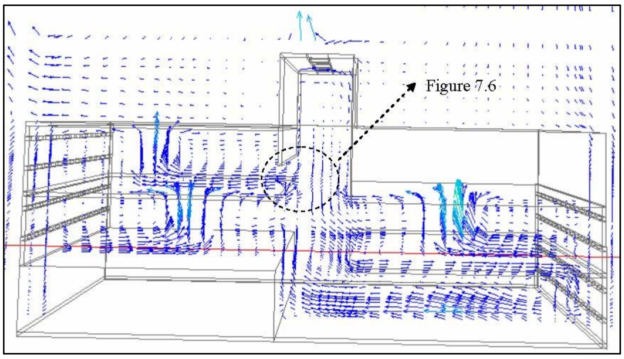


Fig 2. Airflow patterns as calculated with PHOENICS.

When CFD for a single zone is integrated with the multi-node calculation, Dr. Tan found it necessary to zone the building and establish boundary conditions for the CFD. Strategy 1, shown on the left in Fig 3, was less successful than strategy 2 on the right; both produced good temperature estimates but the latter performed better with estimates of air velocity.

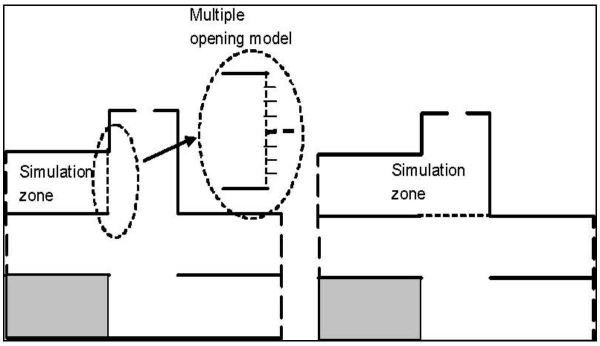


Fig 3. Strategies for combining multi-nodal airflow analysis with CFD simulation of a single zone; strategy 1, on the left, models the zone boundary with eight small openings to permit multi-directional flow; strategy 2, on the right, absorbs the vertical boundary into a single, larger simulation zone that includes the top of the atrium.

Dr. Walker's work centred on constructing the reduced-scale physical model of the naturally ventilated office building in Luton, UK. She and Dr. Tan used the results to validate Dr. Tan's PHOENICS model of the entire building and his hybrid nodal-CFD model. They showed that it was necessary to account for heat conduction through the walls of the CFD simulation in order to match the measured flows and temperatures in the physical model. These heat flows were large because the physical model was operated at high temperatures in order to satisfy similitude relations. In PHOENICS, a negative heat flux was used to mimic conduction.

Dr. Walker also used PHOENICS to compare results in two physical models, using both and water. This was important because we work in collaboration with researchers at Cambridge University who use water models. Both teams attempt to satisfy similitude, which requires that Reynolds numbers exceed the turbulence threshold and that Grashof numbers match what is estimated for full scale; similitude parameters are shown in Table 1. The flow comparison, in Fig 4, showed that the inlet jet due to wind-driven flow did not penetrate the water model as far as in the air model. Thermal stratification also differed.

	Air-Building	Air-Model	Water Model	Water Model
Scale	1	12.	12	100
g (m/s^2)	9.8	9.8	9.8	9.8
β (1/°K)	0.0033	0.0034	0.0002	0.0002
ΔT (°K)	5	30	6	6
g'=gβΔT	0.1655	0.9800	0.0118	0.0118
H= Atrium height (m)	15	12	1.2	0.75
α (m^2/s)	2.16x10^{-5}	2.40 x10^{-5}	1.44 x10^{-7}	1.44x10^{-7}
v (m^2/s)	1.477 x10^{-5}	1.60 x10^{-5}	1.01 x10^{-6}	1.01 x10^{-6}
A_{CS} (m^2)	6.32	0.522	0.559	0.037
Pr	0.7	0.7	7.0	7.0
Re	6.74x10^5	3.54 x10^4	1.10 x10^4	1.70 x10^4
Pe=PrRe	4.72 x10^5	2.47 x10^4	7.68 x10^4	1.19 x10^5
Gr	3.84 x10^{13}	7.94 x10^9	2.05 x10^8	3.65 x10^9
Ra=PrGr	2.69 x10^{13}	5.56 x10^9	1.43 x10^9	2.55 x10^{10}

Table 1. Summary of Values and Dimensionless Parameters for Full-Scale Building, Scaled Air and Scaled Water Models

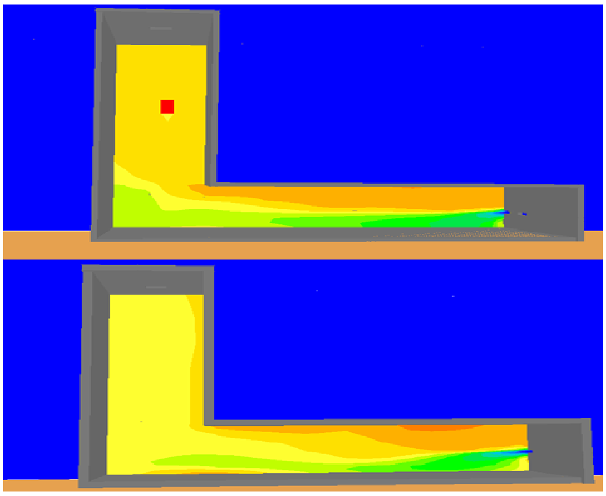


Fig 4. A comparison of reduced-scale air (top) and water (bottom) models

Drs. Walker and Tan investigated combined buoyancy- and wind-driven flows. The circulation pattern showed two-way flow on the leeward sides of the first and second stories, see Figs 5 & 6.

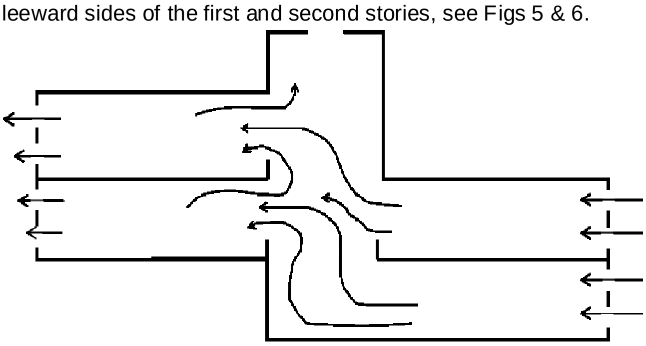


Fig 5. Sketch of airflows under combined buoyancy and wind forces

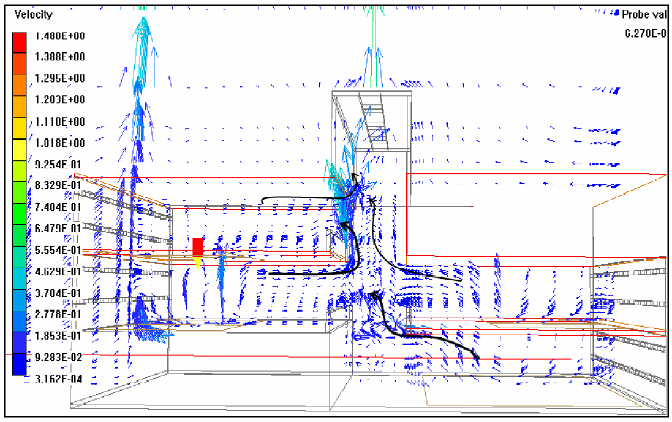


Fig 6. Sectional view of PHOENICS-simulated airflows for buoyancy- and wind-driven flows

Finally, Dr. Walker showed that the atrium railings (clear, solid panels up to about waist height) affect the temperature distribution, as shown in Fig 7. With the railings in place in the simulations, cool air collected at their based rather than mixing with warmer air from the atrium.

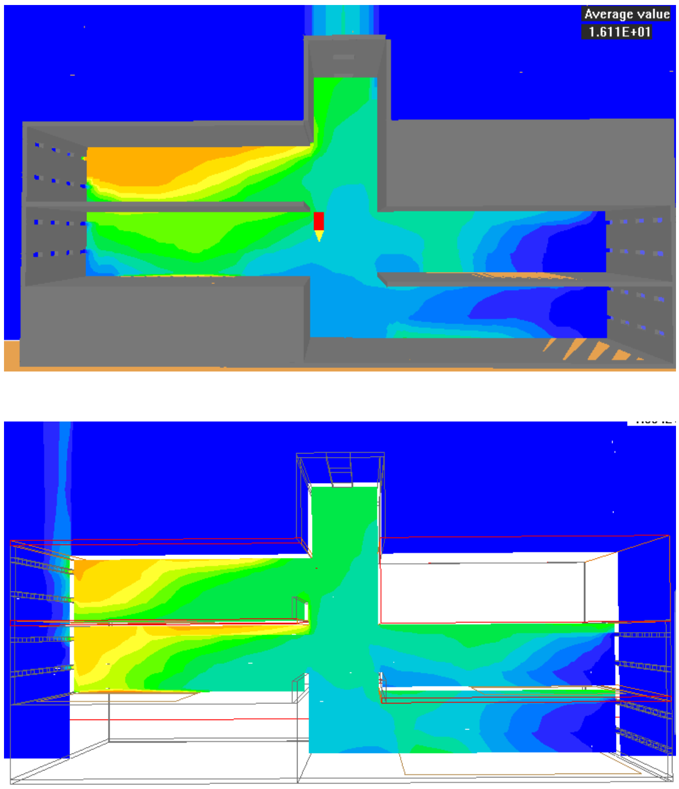


Fig 7. Influence of Railings on Temperature Distribution for Reduced-Scale CFD Simulation; a) without railings, b) with railings

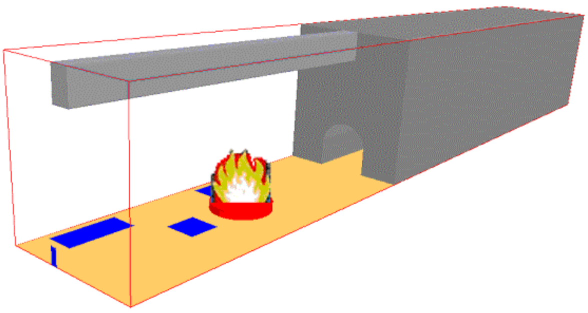
Prof. Glicksman, our graduate students and I continue to be very grateful for CHAM's making PHOENICS available to us. It has been the backbone of our research and has made a significant contribution to our teaching.

Dr Leslie K Norford, Email: lnorford@mit.edu

GEODATA SPRINKLERS EFFECTS ON POWER STATION FIRE

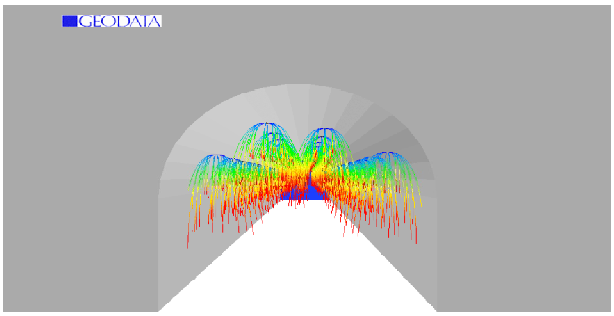
Italian GeoEngineering consultants, GeoData Srl, applied the sprinkler-modelling capabilities of the FLAIR building services module of PHOENICS to a multi-story underground hydraulic power station.

In the scenario described below, one of the two generators is on fire (38MW). All the blue areas are treated as entrances for fresh air from other parts of the complex. There is a walking pass cross the ceiling of the chamber for emergency evacuation. (One might imagine that the original design was for flood instead of fire). The focus of the study is on the tunnel at the end of the chamber.

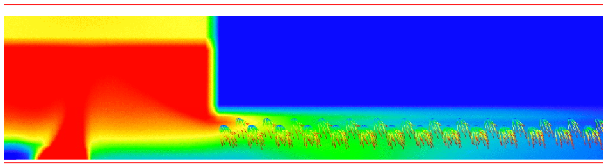


GeoData's client needed to know that the temperature in the tunnel would remain low enough so that the fire brigade could access the complex through the tunnel in which sprinklers are installed for this purpose.

There are 60 sprinklers installed near the entrance of the tunnel.

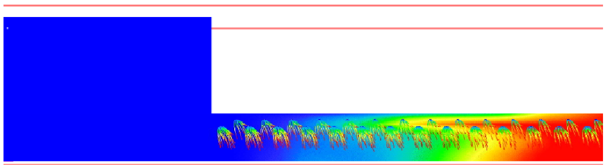


The figure below shows the temperature distribution in the first section of the tunnel.



The temperature range in the access tunnel is between 11°C and 140°C (the temperature in the red region is higher than 140°C).

As expected, the water spray has destroyed the smoke stratification before cooling it down. Unlike a normal tunnel fire in which the general public may be involved, the smoke stratification is not particularly important in this case.



The figure above shows the distribution of relative humidity between 0 and 100%.

Mr Qihui Zhang, Email: gzh@geodata.it

WHAT'S NEW IN PHOENICS-3.6.2

PHOENICS development continues apace both here in the UK and in conjunction with our colleagues at the Science Service Centre in Moscow.

Supervised by Brian Spalding, work associated with fluid/solid interaction, a new unstructured mesh initiative and code parallelisation rests in Russia, whilst the user interface modifications remain the responsibility of the UK. The changes associated with the PHOENICS-3.6.2 update have concentrated on the latter.

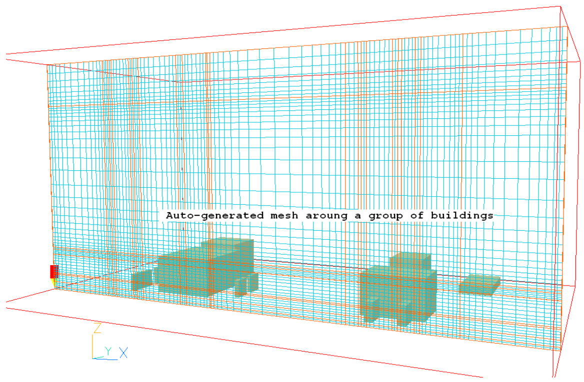
John Ludwig reports on some of these, as follows:

GUI - Common to Editor/Viewer

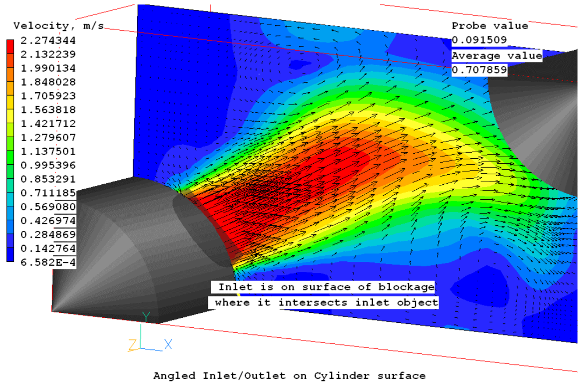
- The handset and movement control panel can be closed, and are replaced by icons on toolbar.
- There is a text facility to annotate plots for reports.
- The axes can be placed anywhere on screen, and will rotate with the domain **Solver**
- Polar PARSOL. The cut cells are calculated for polar geometries.
- Updated SGS turbulence model. Basic (no wall damping) SGS model and Van-Driest wall damping models are added.
- The solution can be written in TECPLOT format.
- Updates to parallel - InForm incorporated more extensively.

Editor

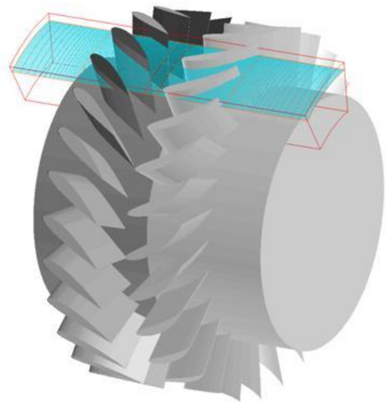
- Auto meshing. The Editor will generate a reasonable grid with little or no user input. This can then be adjusted as required.



- Angled inlet and angled outlet objects. These allow inlets and outlets to be placed on the outer surface of any arbitrarily-shaped blockage.



- Polar update. Non-cuboid geometries, including STL imports, are displayed in polar coordinates with no distortion. The origin is in θ-r-z, but the size is in x-y-z.

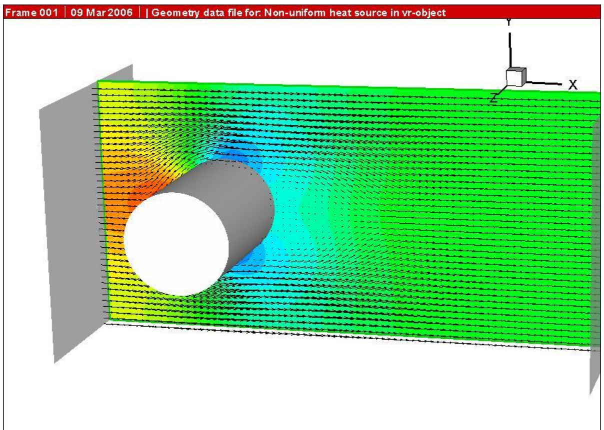


- KE-buoyancy upgrade. A new 'auto' function switches between stable and unstable forms depending on local flow direction, using the function for C3 proposed by Henkes et al.

- MOFOR motion-control files can be created and edited from within the Editor.

- Multiple STL files for import are selected with a file browser dialog (instead of from an externally-generated list file).

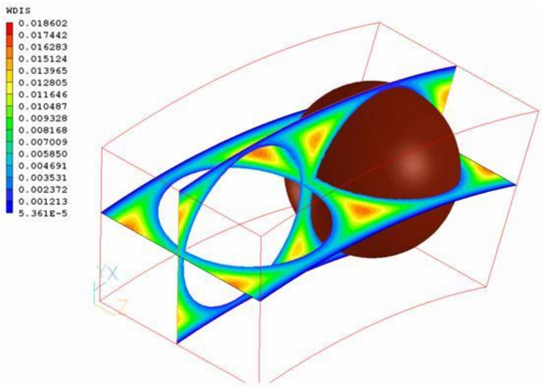
- The geometry can be exported in TECPLOT format. Each object is shown in TECPLOT as a 'zone'. A macro is also written which sets common flags.



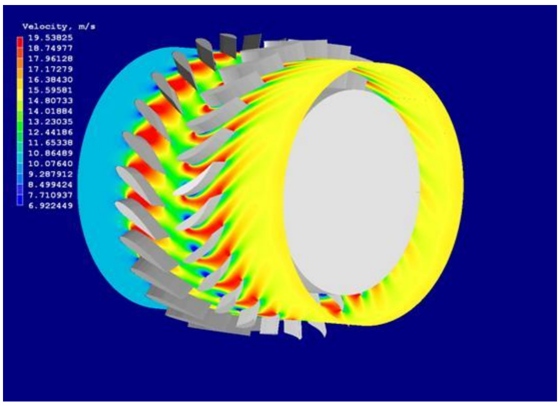
Upgraded PHOENICS-to-TECPLOT interface

Viewer

- Polar PARSOL. Contours and vectors in the cut cells are displayed.



- Output of surface contour and profile values for selected objects to a text file, which can be read into Excel.
- Repeat function for cyclic cases.



- Many internal efficiencies to speed up plotting of large cases.

John Ludwig, Email: jcl@cham.co.uk

CHEMTECH reveals new talents in CFD

During the Science & Technology Week at the Federal University of Minas Gerais (UFMG), 21 graduate students in chemical and mechanical engineering, metals & materials courses took part in the 3rd CFD **Chemtech** / PHOENICS Challenge, a competition of intelligence and skill promoted by CHAM's South American agents, **Chemtech**. The first three places received prizes, both for the students themselves and for their school departments.

"Initiatives such as this not only provide a difference to the students – since they can learn how to use new tools – but also help in valuing the numerical culture", declared Professor Rudolf Huebner, coordinator of the mechanical engineering post-graduation program at the Catholic University of Minas Gerais and a member of the Challenge evaluation committee.

The winning teams of the 2nd & 3rd CFD **Chemtech** / PHOENICS Challenge that took place in São Paulo and Minas Gerais have won much more than just knowledge and prizes. The job market differential that now characterizes the participants is already bearing fruit.

Raphael Chagas, leading student of the runner-up team from UFMG, and Ricardo Próspero, member of the champion team from the São Paulo University, have been admitted as interns at the **Chemtech** offices in Belo Horizonte and São Paulo. Besides these two internship vacancies, **Chemtech** offered training posts for the other Challenge participants during the vacation period.

Chemtech is planning a special edition of the CFD **Chemtech** / PHOENICS Challenge for 2006. The idea is to join in competition, engineering students from all the states of Brazil. Consequently, the chances of employment for the participants will apply to all the **Chemtech** offices, located in Rio de Janeiro, São Paulo, Belo Horizonte, and Salvador.

To learn more about these, and other, Chemtech activities; visit **Chemtech** (www.chemtech.br)

DR JALIL OUAZZANI REPORTS ON WORK UNDERTAKEN BY FRENCH CUSTOMERS ICMCB AND INERIS

ICMCB (Institut de Chimie de la Matière Condensée de Bordeaux) has acquired the PHOENICS code version 3.6.1 to study the behaviour of critical fluids in earth and space environment. The use of near-critical fluids allows important parameters (e.g. compressibility of supercritical fluids, density of gas and liquid phases, surface tension) to be easily varied in a scaled way by using small changes in temperature.

The readily variable properties of near-critical fluids make them appealing candidates for studying numerous interesting phenomena valid for all fluids. Very compressible supercritical fluids such as SF6 in the vicinity of its critical point (45.5°C) can exhibit a very fast transport of heat without convective flows that are not caused by gravity. A hot boundary layer can expand and heat the bulk like a real "piston".

Bulk thermalization has proved to be homogeneous in temperature following a dynamics which is intermediate between typical acoustic times (ms) and diffusive times (hours or days). Taking into account this phenomena is of major importance to understand the phenomenology of critical behaviours. The scientific objective is to study a variety of equilibrium and non-equilibrium phenomena using near-critical fluids (like Sf6).

The topics studied are:

- General physics: critical fluctuations
- Non equilibrium phenomena
- Critical boiling

The numerical aspect is in support of experiments that have been, and will be, held both on Earth and in space, and will help to understand the complex phenomena in such flows.

In the following pictures a cylindrical cell is filled with critical SF6 at 1K of critical point, heated in its centre and vibrated along the axis:

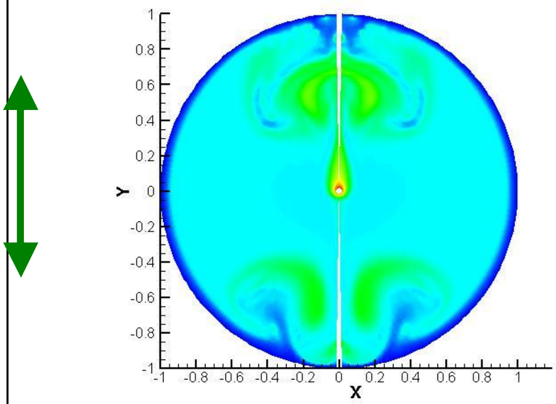


Fig A

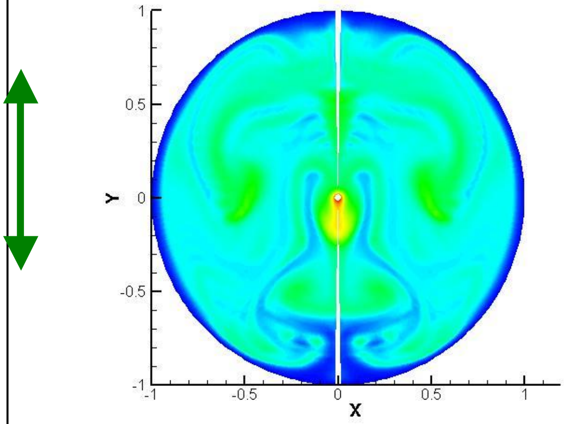


Fig B

Field of iso - density with formation of thermal plumes in the axis of vibration at time t=5s (a), t=10s (b) and t=20s (c). Lower densities are in red colours (corresponding to hot zones) and higher densities in blue colours (corresponding to colder zones).

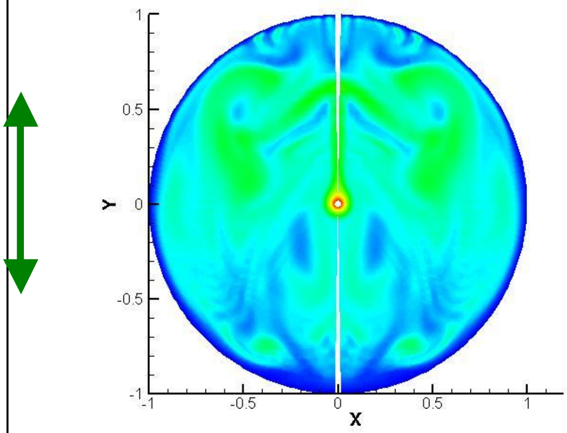
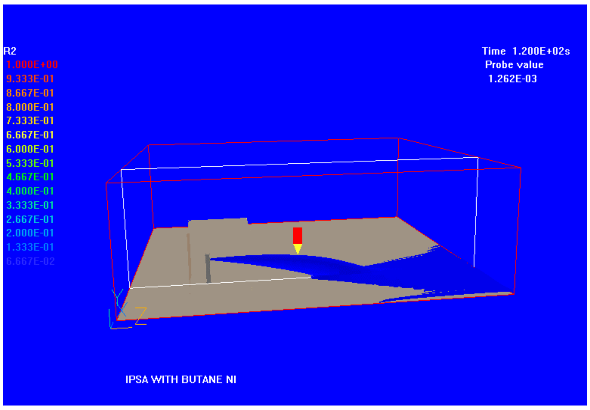


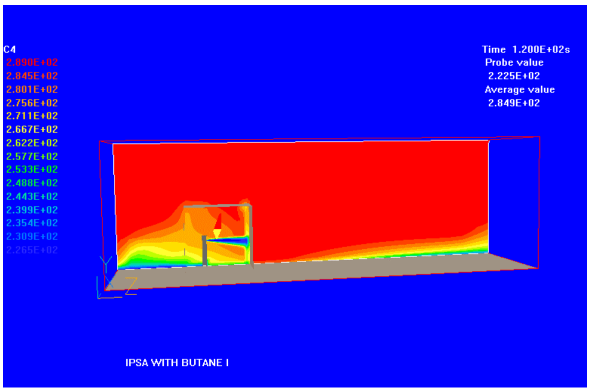
Fig C

The purpose of the study by INERIS was to customize PHOENICS for a framework industrial risk problems concerning the leakage of pollutant or dangerous gases. These were in the form of droplets (evaporating in the ambient air) and vapour, with account taken for surface evaporation from the ground. Two techniques were principally used to address this kind of problem - the IPSA and the GENTRA techniques. These were applied respectively to a case of an impacting jet on a blockage and a case of free jet (not impacting).

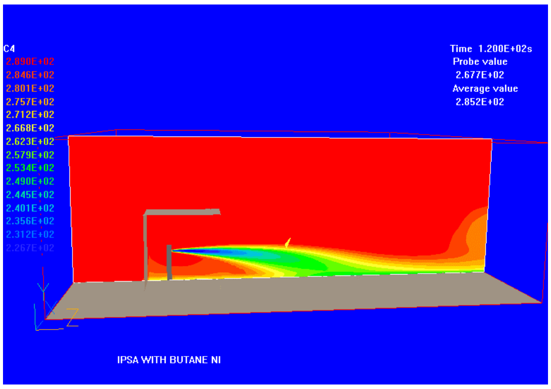
The characteristics of the two cases were identical except that in the case of the impacting jet there is a wall located at three meters of the jet. A jet of butane at -50°C is injected into ambient air at 20m/s. The jet vaporizes from bulk and also from puddle pools. To handle evaporation from puddle pools, specific boundary conditions were introduced.



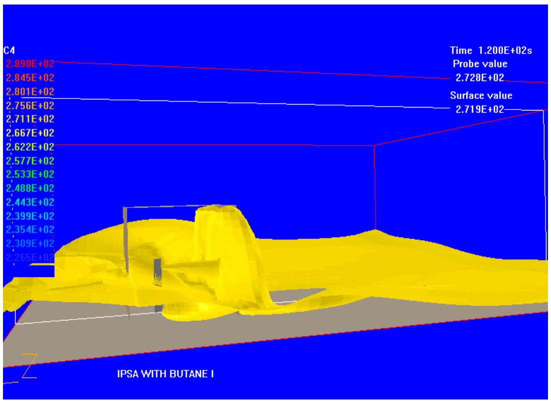
Iso surface (1.262^{E}-03) of volume fraction R2 at time 120s in the case of free jet.



Mean temperature at time 120s in the case of impacting jet



Mean temperature at time 120s in the case of free jet



Iso Temperature (271.9°C) at time 120s in the case of impacting jet

The results compared satisfactorily with experiments conducted at the INERIS site.

Dr Jean Marc Lacome (INERIS, Verneuil en Halatte, France) (Email: Jean-Marc.Lacome@ineris.fr)

Dr Jalil Ouazzani (Arcofluid, Bordeaux, France) (Email: arcofluid@wanadoo.fr)

ASSESSING URBAN AIR QUALITY USING MICROSCALE CFD MODELLING

Currently, most of the local authorities in the UK are using the well-established Gaussian type dispersion models to predict the air quality in urban areas. Although having an enormous potential in assessing and improving natural ventilation in built-up areas, use of CFD in integrated urban air quality modelling is still in its infancy. This study aimed to assess suitability of a general CFD code for use in the integrated urban air-quality modelling for regulatory purposes. By integrating results of traffic flow in urban road networks, traffic emissions, and incorporating a three-dimensional model of a complex configuration of street canyons into a CFD dispersion model (PHOENICS), an urban air-quality model of a designated air quality management area in the city centre of Glasgow has been developed.

To validate the model, the continuous monitoring campaign was carried out in 2002. The experimental data were measured at five different locations in the city centre using both fixed and mobile monitoring stations. The three fixed self-contained monitoring stations were placed at the carefully chosen sites to represent the diversity of Glasgow's micro-environments (urban kerbside, urban centre and urban background locations).

Figure 1. Air Quality Monitoring Trailer

Taking into account the size of the designated air-quality management area in Glasgow, the air pollution was additionally monitored using the automatic monitoring equipment contained in air-conditioned trailers (Figure 1). The locations were selected according to the traffic-flow data in order to give more detailed information on local air pollution differences within the street canyons in the designated air-quality management area.

A comprehensive preliminary CFD study conducted was aimed at highlighting the fundamental problems of micro scale CFD models, which lie in the physical difficulties of modelling the effect of turbulence, and also the accuracy of the spatial discretisation of complex urban geometries, the numerical procedures applied, the boundary conditions and the physical property selected.

All conclusions of the study were underpinned by a wind tunnel and real-scale experimental data. A guideline on use of PHOENICS in airflow studies in complex built environments was developed.

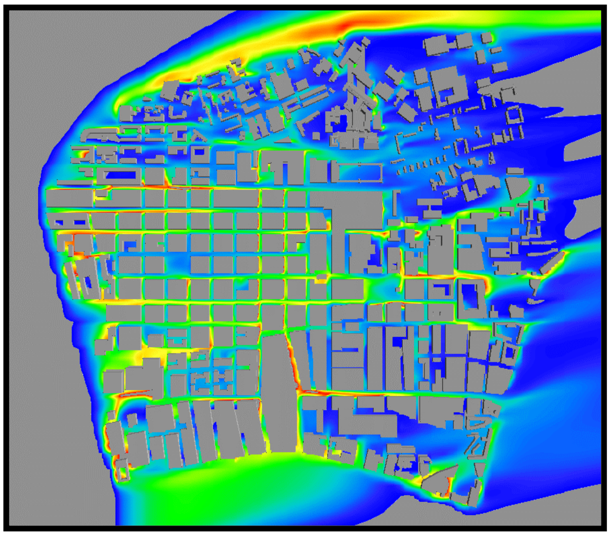


Figure 2. Glasgow – modelled CO levels

The model developed (Figure 2) shows absolute consistency, 100 %, in over-prediction of field results within the street canyons where the wind direction is almost perpendicular to the street axis. It has to be noted, if analysing separately, that the model shows very high consistency (90 %) in over-predicting the concentration when the prevailing winds are southwesterly. On the other side, due to alignment of the computational cells with the westerly winds, the model gives encouragingly accurate results with the calculated relative error of 25 % on average. Note that the modelling quality objectives are set by European Union Directive 2000/69/EC, and the recommended value for 8 hour running mean, and one hour averaged concentration of carbon monoxide is 50 %, and 60 % consequently.

Although this numerical tool demonstrated satisfactory performance, it was observed that small differences in monitoring station positioning might yield significant variations of measured mean concentration due to large values of horizontal and vertical local concentration gradients. Although, at this stage, the accuracy of developed Glasgow urban air quality model highly depends on experience of its users, it is believed that use of CFD in an integrated urban air quality modelling could be to the benefit of urban planners, architects, HVAC engineers and all other professionals interested in public health.

Dejan Mumovic, Complex Built Environment Systems Group, The Bartlett, University College London, Email: d.mumovic@ucl.ac.uk

Zarko Stevanovic, Institute of Nuclear Sciences VINCA

John Crowther, School of Built and Natural Environment, Glasgow Caledonian University

GATEWAYS TO PHOENICS

New “Gateways” to PHOENICS have been designed to make it easy for persons who wish to simulate fluid flow in situations of narrowly-defined types to do so without having their attention drawn to matters which do not concern them, or needing to know anything about PHOENICS in general.

Individual gateways are accessed by clicking one of the icons presented on the left of the main-gateway panel, whereupon further icons will appear which offer simulations of particular kinds. Clicking on these will load the selected generic input file into an interactive software module called “PRELUDE”, which enables its user to:

- inspect both the items on offer, and the geometrical and other specified relations between them [The REL in pRELude refers to 'RELations' which are PRELUDE's reason for existence];
- modify the items provided, including both their attributes and their relations;
- delete items or introduce new ones:
- when satisfied with the resulting scenario, launch one or more flow-simulating operations, and
- thereafter inspect the results.

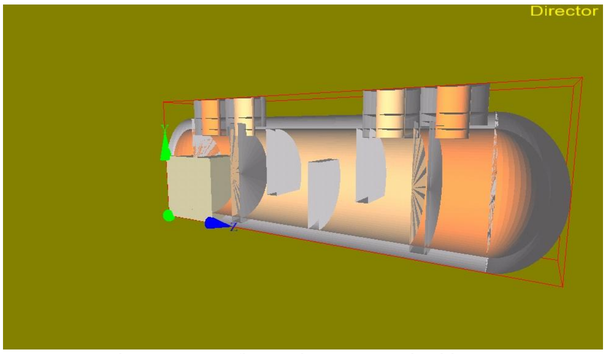
In designing a gateway therefore, CHAM seeks to identify a group of potential users having similar needs; define those needs as precisely as possible; and prepare for them:

1. a starting scene which already contains the items which a typical user will need;

2. a 'store-cupboard' stocked with such other items as the user may wish to add; and

3. a ‘wizard’ that will assist the user to:
- add and remove items,
- adjust their positions and other attributes,
- conduct the flow-simulating calculation, and
- inspect the results.

Persons who are familiar with PHOENICS may be interested to know that PRELUDE can be regarded as a pre-pre-processor, in that it delivers its output to the standard SATELLITE, which then converts it into EARTH-usable form.



Heat exchanger configuration created with PRELUDE

PRELUDE does both more and less than the SATELLITE. More, in that it works with an input file, like a 'parameterized Q1', which can express user-definable relationships between input variables; and that it can launch a series of runs with varied inputs. Less, in that it facilitates the user's access to only the limited set of PHOENICS features that the input-file writer has believed to be necessary.

- PRELUDE has its own graphics package, based on the OpenSceneGraph library.
- PRELUDE is written in TclTk, and is therefore platform-independent.
- The input file (at the present moment called a Q3 file) can be hand-edited. The Q3 contains (and retains) all generic features of the class of scenarios; but it can generate as many single-instance Q1 files as its user decides.
- Although PRELUDE, operating in 'gateway' mode draws the user's attention only to those features that the gateway-designer has selected, it does not diminish the user's freedom to activate any other feature of PHOENICS.

CHAM's introduction of PRELUDE also expresses its new approach to the market place. In the past, the developers of PHOENICS have furnished it with many valuable features, some of which are described in 'what's new' announcements or newsletters, and fewer of which have been illustrated in single-instance examples added to the input-file library.

PRELUDE has been designed to enable CHAM itself to create the ‘gateways’, which are described in this document, and of which the essential feature is their users require no knowledge of PHOENICS, or even of computational fluid dynamics.

Brian Spalding, email: brianspalding@cham.co.uk

CHAM CONSULTANCY UPDATE

During 2004 and 2005, CHAM was engaged by Epsom-based Atkins Water to study a series of effluent releases into river systems. Most recently, a study was undertaken to predict localised effects of storm-water outfall into a tidal region of the Herdman Channel in Belfast, Northern Ireland.

Site of storm-water outfall

The client was particularly concerned about the potential erosion of the estuary's riverbed and banks during high loading conditions. The CFD model took into consideration different outfall and estuarial flow rates. Studies were undertaken to model worst-case scenarios for low and high tide conditions, and also to establish the likelihood of any adverse effect on river traffic. An Atkins Project Experience Sheet reports:

*"**Belfast Outfall, Northern Ireland - Belfast Harbor Commissioners***

Increasingly, Computational Fluid Dynamics [CFD] is playing a part in engineering hydraulics, particularly in situations where the flow is complex and highly three-dimensional. Atkins was commissioned to provide a design for an outfall in Belfast Harbor. We applied CFD in order to assist the design.

This was part of a major scheme to deal with storm water in the city of Belfast. The proposal outlined the plan to discharge storm water flows into the Herdman Channel in Belfast Harbor. By discharging into the Herdman Channel rather than nearer the coast, a considerable amount of construction and cost can be avoided. The maximum flow of 16cumecs is large enough to pose a number of potential problems such as: -

1. *The jet emanating from the diffuser might constitute a navigational hazard to shipping maneuvering within the channel.*
2. *The discharges could undermine a local sheet pile pier structure due to scour or erosion.*
3. *The bed shear stress could affect sedimentation patterns.*
4. *The diffuser must provide adequate dissipation of energy from the flow in the culverts upstream of the diffuser structure.*
5. *The diffuser must provide adequate dilution of the effluent.*

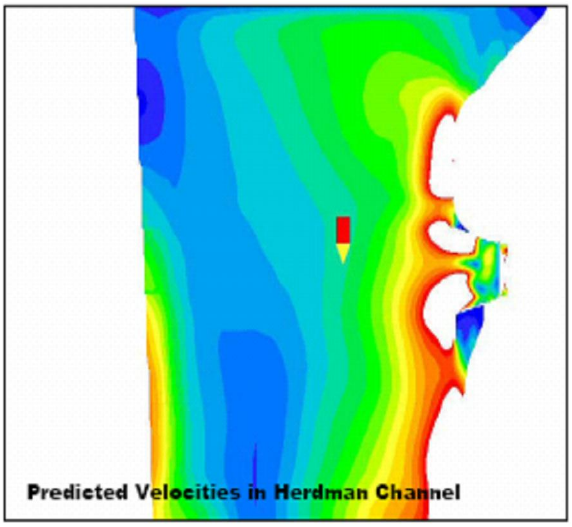
To mitigate against such problems, an outfall diffuser structure was developed. Its performance against the 5 criteria detailed above was tested using PHOENICS.

The modelling showed that: -

- *There was no navigational hazard.*
- *Easily constructed wing walls and an apron would avoid local erosion.*
- *Sedimentation in the channel would be minimally affected.*
- *Dilution was poor but only for very infrequent storm events.*

Overall the design was judged satisfactory so that tunnelling to the coast and a more remote outfall could be avoided, thus saving considerable resources.

The project has therefore benefited considerably from the application of CFD to an environmental hydraulic engineering problem."



George Mitchell, Principal Environmental Modeller
Email: george.mitchell@atkinsglobal.com

NEWS & EVENTS

Professor Brian Spalding will be giving the keynote speech in memory of Professor Jim Hartnett at IHTC-13, the 13th INTERNATIONAL HEAT TRANSFER CONFERENCE to be held in Sydney, Australia from 13th to 18th August, 2006
Email: ihtc-13@unsw.edu.au
URL: http://www.ihtc-13.com

Training Courses – the next 3-day training courses will be held at CHAM from:

21st to 23rd March 2006
23rd to 25th May 2006
27th to 29th June 2006

Contact Sales@cham.co.uk to reserve a place.


Snapshot from a PHOENICS seminar held in Beijing in February 2006 – a similar, equally successful, event was held in Shanghai during March.

AGENTS NEWS

The ARBS (Air Conditioning, Refrigeration and Building Services) 2006 Exhibition in Sydney Australia being held at Darling Harbour in conjunction with the 13th International Heat Transfer Conference from 14th - 16th August 2006.

PHOENICS will be demonstrated at the exhibition along with other software that ACADS-BSG distributes. Such events provide excellent opportunities for people interested in CFD modelling to find out more about the capabilities of the CHAM software.

For further information, please visit http://iir-irhace2006.org.nz, www.arbs.com.au, www.ihtc-13.com or contact the ACADS-BSG office.

Email: acadsbsg@ozemail.com.au

Norwegian agent LM-FlowConsult has merged with the Swedish group company ÅF. The name of the Norwegian part of the company is **ÅF-Consult AS** from January 2006.

As a result of the new arrangement, PHOENICS has been licensed to various ÅF sites including ÅF-Process AB in Stockholm, ÅF Helsingborg and ÅF Varberg – with others to follow.

The company is continuously growing, and has about 3100 employees by now. ÅF has just bought Enprima Ltd in Finland. You can read about it on their web site:
http://www.afconsult.com/index.asp?id=2565

WindSim User Meeting in Tønsberg 18 - 20 June Share your experience and get updates on the most recent WindSim developments. For information and registration please visit:
http://windsim.com/events/courses/index.html

Tine Volstad, Email: tine@windsim.com

We say a sad good-bye to Hans Mindt of German agents **a-CFD** with whom we have worked for many years. We send our best wishes for his future in his new venture with the ESI-Group. Losing Hans means that there is a temporary void in our sales/support representation for Germany that will be filled by the UK and Dutch agencies for the interim, until a new agent is appointed.

PHOENICS-using consultants, **PCM-Thermal Solutions Inc**, now represents PHOENICS in the Mid-West of the USA. PCM-Thermal, and its associate company, MJM Engineering, specialise in electronics and telecom cooling, and thermal management technologies.
Email: sales@pcm-solutions.com
Web: www.pcm-solutions.com

We are also pleased to announce the appointment of CFD consulting firm, **ACS Consulting Inc**, as new agents for the Mid-East of the USA. Until recently ACS Consulting was an independent and utilized a variety of codes, but changed its marketing strategy in the light of new opportunities offered by PHOENICS.

ACS Consulting offers a broad spectrum of CFD expertise and will, in due course, offer training and technical support services for PHOENICS in addition to specialist assistance for parallel-processing implementations.
Email: cfdconsu@cfd-consulting.org
Web: www.cfd-consulting.org

CORPORATE ADDRESSES

CHAM Limited
40 High Street, Wimbledon Village
London SW19 5AU, United Kingdom
Tel: +44 (0)20 8947 7651
Fax:+44 (0)20 8879 3497
Email: phoenics@cham.co.uk
Web: www.cham.co.uk

CHAM Japan
406 Kojimachi Intelegent Bldg 4F, 3-5-4 Kojimachi,
Chiyoda-ku, Tokyo 102, Japan
Tel: +81 35 21 09 356
Fax: +81 35 21 09 359
Email: customer@phoenics.co.jp
Web: www.phoenics.co.jp

AGENT ADDRESSES

AUSTRALIA - ACADS-BSG
email: acadsbsg@ozemail.com.au
web: www.ozemail.com.au/~acadsbsg

BALKANS - Vinca Institute of Nuclear Sciences
email: xgrafon@eunet.yu
web: www.vin.bg.ac.yu/140

BRAZIL – CHEMTECH
email: chemtech@chemtech.com.br
web: www.chemtech.com.br

CHINA - CHAM Shanghai
email: cham@phoenics.cn
web: www.phoenics.cn

EGYPT - Hydro Consult
email: elbaz@link.net

FINLAND - GEM Systems
email: gem.systems@gemsystems.fi
web: www.gemsystems.fi

FRANCE – Arcofluid
email: arcofluid@wanadoo.fr
web: www.arcofluid.monsite.wanadoo.fr

GREECE – Sybilla
email: mail@sybilla.gr
web: www.sybilla.gr

INDIA - C-DAC
email: vikask@cdacindia.com
web: www.cdac.in

INDIA - KLG SYSTEL LTD
email: navendu@klgsystel.com
web: www.klgsystel.com

ITALY - Lasertec SRL
email: l.vittori@lasertecsrl.it
web: www.lasertecsrl.it

JAPAN - CRC Solutions
email: a-suzuki.crc.co.jp
web: www.crc.co.jp

KOREA - ACT2000
email: dsyeom@act-e.co.kr
web: www.act-e.co.kr

MALAYSIA – Cybron Technology
email: danalakshmi@cybron.co.my
web: www.cybronasia.com

MEXICO - Vortex de Mexico
email: vortexmx@prodigy.net.mx

MIDDLE EAST - Locke Carey Consultants
email: lcfsc@emirates.ae
web: www.lockecarey.com

NETHERLANDS – A2TE
email: gjannsen@a2te.nl.com
web : www.a2te.nl

NORWAY - ÅF-Consult AS
email: terje.toften@afconsult.com
web: www.afconsult.com/no

SPAIN - Aertia Software SL
email: jmsole@aertia.com
web: www.aertia.com

TAIWAN – CHAMPION
email: sales@c-h-a-m-p-i-o-n.com.tw
web: www.c-h-a-m-p-i-o-n.com.tw

TURKEY - Hidronerji ve Enerji Sistemleri, Muhendislik
email: ytopcu@hidronerji.com
web: http://hidronerji.com

UK - S&C Thermofluids
email: tony.smith@thermofluids.co.uk
web: www.thermofluids.co.uk

UK - Flowsolve Ltd
email: cfd@flowsolve.com
web: www.flowsolve.com

UKRAINE - IET, NAS
email: ksg@vortex.org.ua
Web: www.ittf.kiev.ua

USA / CANADA – ACFDA
email: acfda@sympatico.ca
web: www3.sympatico.ca/acfda

USA – ACS Consulting Inc
email: cfdconsu@cfd-consulting.org
web: www.cfd-consulting.org

USA - DD&G Inc
email: parisp@bellsouth.net

USA – PCM Thermal Solutions Inc
email: sales@pcm-solutions.com
web: www.pcm-solutions.com

First for the photo gallery !

Dr Vladimir Agranat
ACFDA