Summer / Autumn 2015

Modular Structure of PHOENICS-2015, Brian Spalding, CHAM

Simulation scenarios, defined by special-purpose parameterised Q1s, *e.g.* as below:
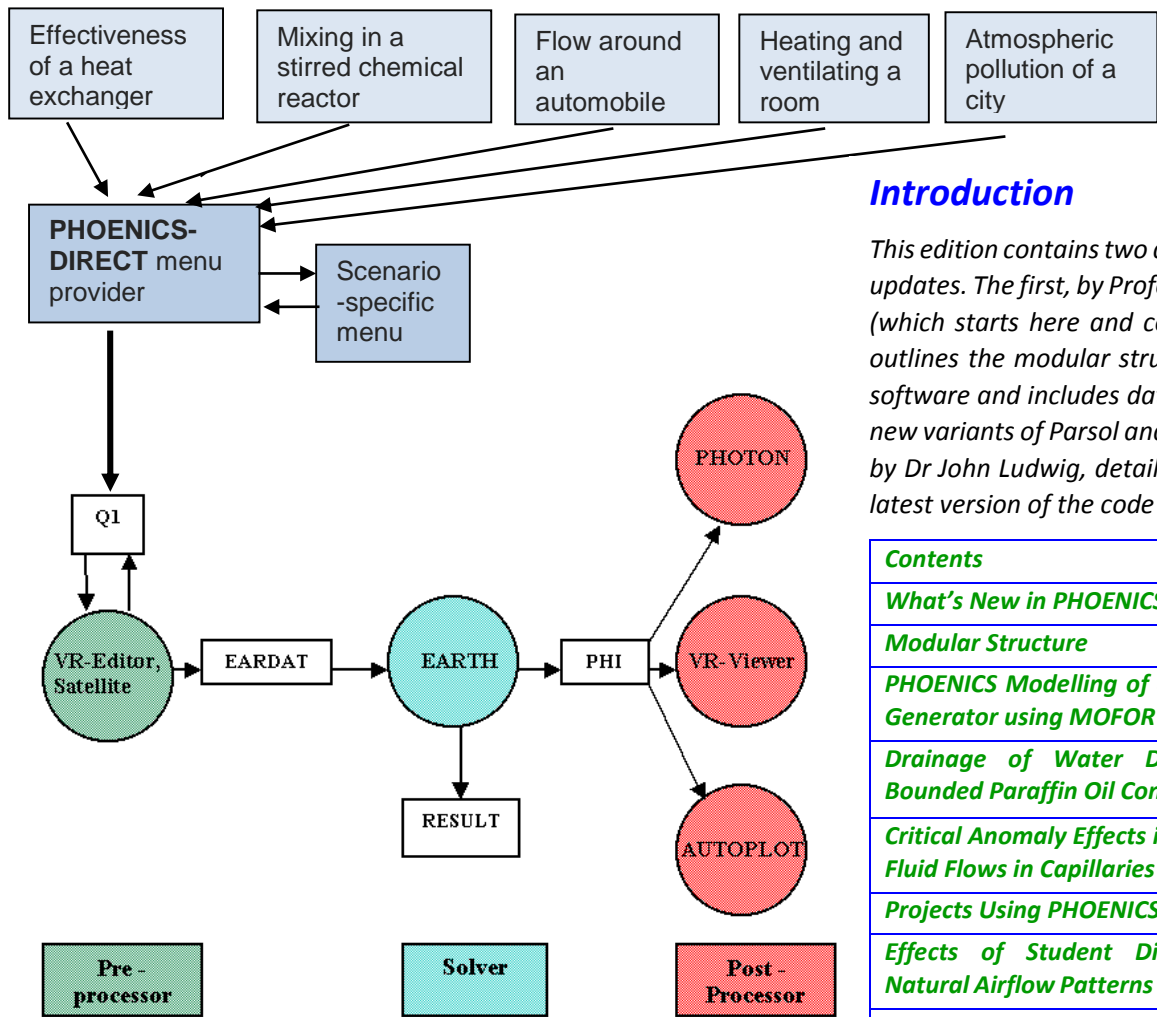
Introduction

This edition contains two articles on PHOENICS updates. The first, by Professor Brian Spalding, (which starts here and continues on page 2) outlines the modular structure of the current software and includes data on SimScenes and new variants of Parsol and Spinto. The second, by Dr John Ludwig, details what is new in the latest version of the code – PHOENICS-2015.

The new modular structure

The lower half of the previous diagram represents the structure of PHOENICS that has prevailed for many years. PHOENICS 2015 contains the same features, working as before or better. The upper part of the diagram reveals that it also contains an ever-increasing array of new modules.

These Simulation Scenarios are being created in response to the recognition that most prospective beneficiaries of Computational Fluid Dynamics have no time to master an interface as easy as we thought the Virtual Reality Editor to be. They need menus presenting questions only about their specific applications, here the answers are known.

The special-purpose-program idea is not new to PHOENICS, as witness FLAIR, HOTBOX, CORA and others; but in the past it has involved special Fortran coding, in EARTH or Satellite, with the inevitable difficulty and expense of maintenance.

Simulation Scenarios do not have this requirement; all their defining features are written in the long-existing PHOENICS input language, (PIL) which facilitates speedy implementation.

Underlying each SimScene (as they are called, for brevity) is a parameterised Q1 file; but users need know nothing of it; they see only the menus enabling them to set values of the parameters which they need.

There are already many SimScenes in existence or under construction; and as many more may be expected to appear as there are CFD-application sectors in engineering and the man-made or natural environments.

Therefore only a few are being provided as part of the PHOENICS-2015 package to Users with standard Maintenance agreements. Those few are of a tutorial character, which convey the general idea and provide information about what others are or will be available

The heat-exchanger tutorial

The image below shows the opening screen of one of the SimScenes supplied. The buttons along the top perform functions which are the same for all SimScenes. The text beneath is the top page of the document specific to the Heatex-Tutorial SimScene. The **structure** of that document is again general, uniformity of format being one of the working practices which make new SimScenes quick to create.
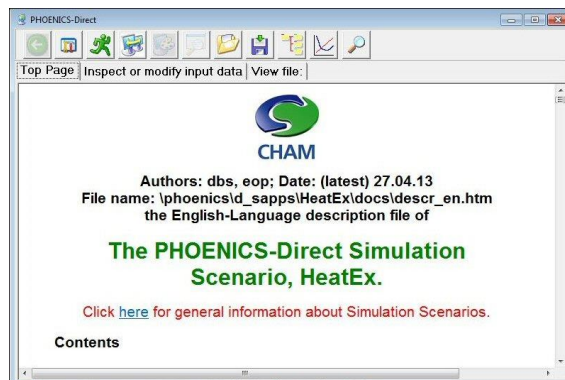

The next image shows one of the menu items PHOENICS-Direct generates automatically when the Inspect-or-modify button is pressed. Its format is general. Only parameters vary from SimScene to SimScene. Those shown here are meaningful to heat-exchanger specialists.
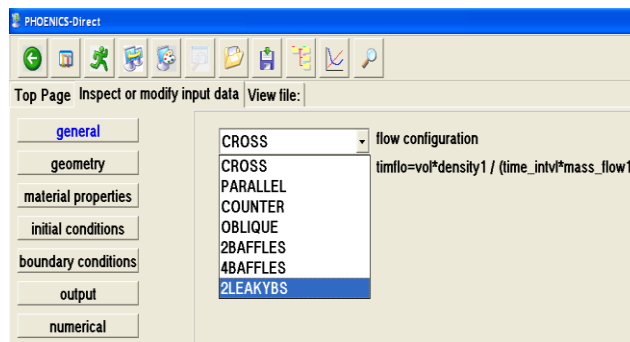
Clicking the green running man executes the simulation; macros provide commands which elicit

graphical displays. That below represents shell-side (top) and tube-side (bottom) temperatures in a time-dependent calculation. Top-left is early; bottom-right is later.
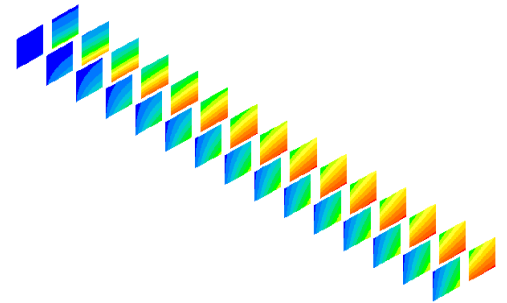

SimScenes provide more insight then any text-book can; for they take into account factors such as temperature-dependent properties and buoyancy effects which are beyond the scope of analytical methods. Interactive graphical display of result permits easy exploration of cause and effect. These facts are of course known to all CFD experts; but it is the existence of SimScenes which enables non-experts to understand them too.

The power of PIL

CHAM has not needed to make massive developments in order to bring SimScenes into existence; exploiting and somewhat enlarging the already-existing capabilities of PIL, the PHOENICS Input Language, has sufficed.

PIL has existed since PHOENICS began; but, while the Virtual Reality Editor was the favoured interface, its power was seldom exploited.

With PHOENICS 2015, there will be provided the first of a to-be-enlarged set of so-called 'PIL Objects' which supplement those that have long been in the Satellite's d_object directory. Two are shown below.
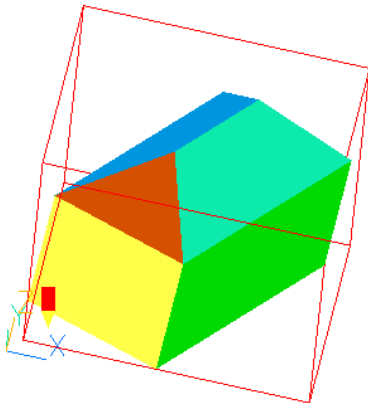
HOUSE.dat created by q1

The image above represents a house, the few geometric parameters of which are set in PHOENICS input file, Q1. Experienced users of earlier PHOENICS versions might have been able to construct it by laboriously putting together a set of 'cube' and 'wedge' objects, which exist in the d_object directory; now the user has simply to type parameters into menu boxes. If he wishes to provide a complete housing estate, the do-loop capability of PIL makes that easy.

The image below represents a circular ramp such as is often found in car parks. Previously it would have had to be created using a CAD package, with its output converted to a format acceptable to PHOENICS. PHOENICS-2015 users can supply defining parameters to the Q1 file and run Satellite. Much time is saved thereby.
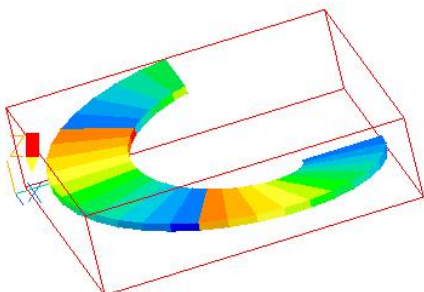
CIRCRAMP.dat created by q1

The PIL-object concept is too new for many exemplars to be in the current delivery version. If others are desired, users who understand PIL can create them. Otherwise, CHAM will do so free of charge - if they are judged to be of general interest.

Other novelties

Variants of PARSOL.

PHOENICS has, for many years, possessed the PARSOL feature which enables flows around bodies of arbitrary shape to be computed on structured grids. The near-surface cells of such grids have been split into two. One part contained solid and the other fluid (hence the name). However, as at first implemented **only two** parts were allowed. This meant that thin objects could be allowed only with very fine grids; which entailed sometimes burdensome expense.

The first remedy, released with PHOENICS 2014 and known as SPARSOL, eliminated the necessity for doubly-cut cells by changing the sizes of nearby un-cut cells. It enabled arbitrarily thin objects to be handled, successfully (as described in the PHOENICS Newsletter Summer 2014).

This option is retained in PHOENICS 2015, which is also being equipped with a further remedy, known as XPARSOL.

This allows the doubly-cut cells to exist; and deals with the problem of the increased number of dependent variables by modifications to the solver which now deals with one set of fluid cells on odd-numbered sweeps and the other on even-numbered sweeps.

Because XPARSOL is new, and therefore less tested, PHOENICS 2015 is being released now with SPARSOL and, later, with XPARSOL.

Users may decide whether to use SPARSOL, the default, or switch to XPARSOL in due course. Solutions provided by the latter are expected to be marginally more accurate; but experience is not yet abundant.

The odd-even solution device has been recognised as having merits for

other applications. Free-surface flows, and simultaneous fluid flow and solid stress are among them, as future versions of PHOENICS will reveal. The development journey begun in 1981 is still not ended.

SPINTO

SPINTO, the **S**implified **P**HOENICS **INT**erp**Olator** has been upgraded and given increased prominence in PHOENICS 2015. Its purposes have been clarified as being two-fold:

- SPINTO-TS is used for **T**ime **S**aving; and
- SPINTO-MA is used for **M**aximising **A**ccuracy.

In its **time-saving** mode, SPINTO solves fine-grid problems by way of a succession of runs, the first being on a coarse grid, and subsequent ones on finer and finer ones. The essential feature is that each run derives its initial-value field by interpolation in the solution field of its predecessor.

Provided that judicious run-termination criteria are adopted for earlier runs in the sequence, total run time to secure a prescribed accuracy of finest-grid solution can be appreciable. It should be noted **that new run-termination and accuracy-assessment** methods are provided in PHOENICS 2015.

All SPINTO-TS runs are performed with the same domain size and boundary conditions. Only grid-cell number and distribution change.

In its **maximising-accuracy** mode, by contrast, domain-size and boundary conditions which change, from run to run, while cell number and distribution may remain the same.

SPINTO-MA is used when, with the finest economically permissible grid, the accuracy of the solution in the region of special interest remains in doubt. The assumption is then that the solution outside that region is

good enough; therefore. It can be used to provide initial and boundary conditions of a subsequent run in which the domain is smaller than before, namely the region of interest. Having the same total number of cells, this fits the therein-contained objects more closely.

SPINTO-MA can be operated in two distinct manners, namely 'one-way' or 'two-way'. In the former runs focus on ever smaller regions and then stop. In the latter, recognising that a larger-domain solution would have been different if it had known what the next-smaller-domain solution later reveals, the latter information is 'fed back' to the larger domain in the form of volumetric mass, momentum and energy and energy coefficients. A new larger-domain solution is then obtained; and a new (and hopefully more accurate) smaller-domain solution is procured.

Another new SPINTO feature in PHOENICS-2015 is that, whilst a stand-alone SPINTO module still exists, an inside-EARTH-coding implementation of the interpolation process exists also. This, and the associated SimScene, makes SPINTO much easier to use than before.

Connected multi-runs

The SPINTO operations described have in common the fact that data input to later runs is influenced by data output by earlier runs. They are therefore instances of the class of 'Connected Multi-Runs', of which other instances are:

- two-stage simulations of heat-exchangers, in which complete-equipment simulations employ space-averaged representations of tube-bank friction and heat-transfer coefficients. The coefficients are not, as formerly, based on doubtfully-adequate extrapolations of experimental data, but on detailed-

geometry CFD simulations of volumes containing a few accurately-represented tubes. The two simulations interact cyclically until convergence.

- two-stage simulations of car-parks, in which important but relatively small items such as curved ramps (see above) are represented by volumetric coefficients drawn from detailed-geometry simulations of single ramps, with boundary conditions imported from the whole-car-park simulation.

- two-stage simulations of aircraft in which solutions of the elliptic equations of flow outside the boundary layer exchange information iteratively with solutions of parabolic equations which describe behaviour of flow within the boundary layer

The main advantage of these two-stage strategies is that their solutions are more accurate, and economically obtained, than when a single-stage solution with a fine-as- necessary grid is used.

FINAL WORDS:

The changing world of CFD

In 1981, PHOENICS set a pattern which others swiftly copied, from Fluent, through Star-CD to Open-Foam.

Each was a single do-it-all code using a single contain-it-all grid. In CHAM's view, the role of such codes is changing in response to users' demands and development of more sophisticated solution techniques.

Users, being more numerous and more varied in knowledge and needs, require a wide range of tailor-made easy-to-use front ends.

CHAM's PHOENICS-Direct, and uniform-format but user-specific content, allow these to be economically provided.

Even with the largest computers available to most CFD users the contain-it-all grid is seldom fine enough for reliable accuracy. The automatically-organized two- (or multi-) stage approach offers the desired level of accuracy more affordably. Only CHAM, among major CFD code providers, appears to recognize this at the present time.

So that users may explore the influence of grid size, CHAM is providing tutorial SimScenes in which commonly-encountered situations are constructed from parameterised PIL objects. The following images relate to flow through a vehicle ramp such as is found in car-parks.
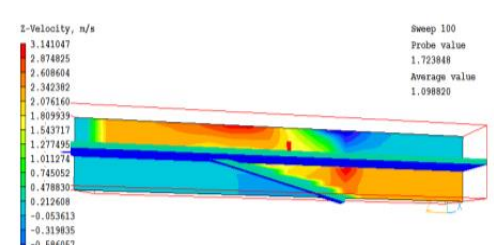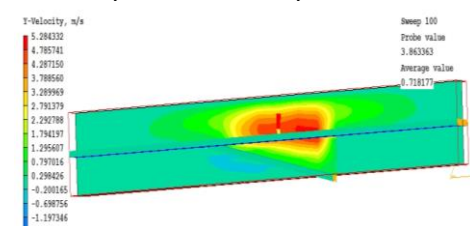
First velocity vectors:
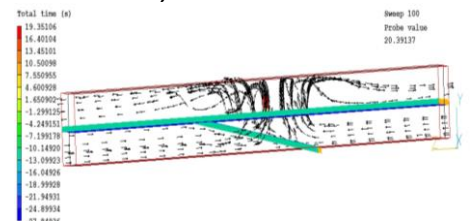
then pressure distribution on the central plane:
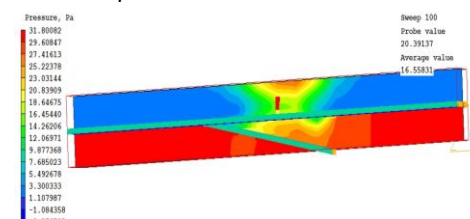
followed by horizontal-velocity distribution:
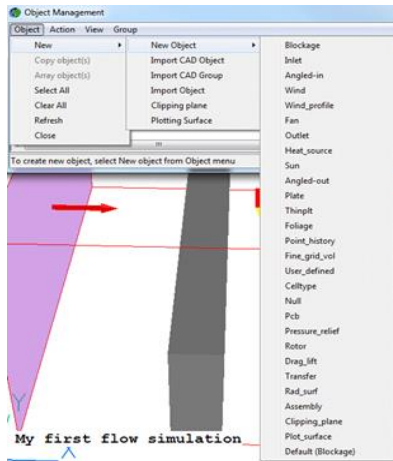

and, lastly, vertical-velocity distribution:

A PHOENICS-Direct menu enables grid and geometry to be changed easily.

What's New in PHOENICS 2015, *John Ludwig, CHAM*

New Editor Features

Object Creation

Object / New / New Object selected right away. Before, to create a new object the default object type was BLOCKAGE, which often had to be changed to the required type.


WIND Object

The WIND object can take inlet profiles of velocity and turbulent kinetic energy from files containing tables of velocity or turbulent kinetic energy against height.

The image shows a tabular inlet profile entering the domain over a wedge representing the terrain.

The WIND object initial velocity log-law profiles are fully compatible with in-Earth inlet profiles by adding tests on h < z0 and WALL ≠ B0.
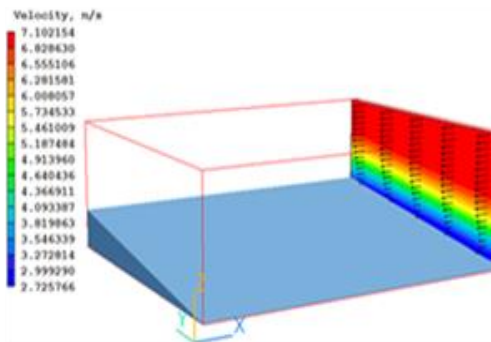
PLATE Object

In a transient case, the PLATE now operates fully within set time limits – the porosity is also switched off (set to 1.0) outside time limits e.g. to represent opening / closing door. Previously only the sources attached to the PLATE obeyed the time limits, the porosity was always present.

CAD Import

The maximum number of CAD files selectable at one time is now a CHAM.INI item, defaulted to 500. Should a user wish to import more than 500 files, they need to change MAXCAD in the CHAM.INI file.
[Satellite]MAXCAD = 500
The import of CAD files no longer fails if there is more than one '.' in the file name, e.g. 'part.1.3ds'.

Mesh Generator

The graphics screen refreshes when 'Apply' is clicked in the grid menu making it quicker and easier to see effects of changing grid parameters.

Miscellaneous

Patches with no COVALs are deleted before writing the eardat file making eardat files smaller and reducing 'clutter' in the RESULT file echo.

Echoing of ANGLED_OUT objects to Q1 in two phase cases corrected. Phase-2 pressure coefficient not written.

NX/NY/NZ greater than 999 allowed when writing GCV multi-block link.

Write Q1 for SEM echo heavy fluid at inlet / outlet as 'HEAVY' not 1.0.

Covals at SEM inlet corrected to account for volume flow, mass flow or Cartesian components

InForm linkage between object and patches corrected.

PIL

The new PIL command:
    SPLIT(arg1,arg2,arg3)
will take a character string **arg3**, split it into individual words using ',' as separators and then place each word into elements of PIL array **arg1**.

Arg2 returns number of words found, and dimension of the **arg1** array. If **arg1** does not exist, it is created.

The new PIL command OBJINFO will write details about a selected object, or all objects, into a designated file.

Reading of the Q2 file is delayed until just before writing eardat so that all object information is present. This allows an OBJINFO command in Q2 to return details of all objects.
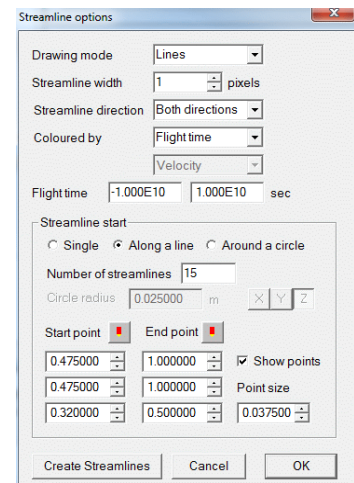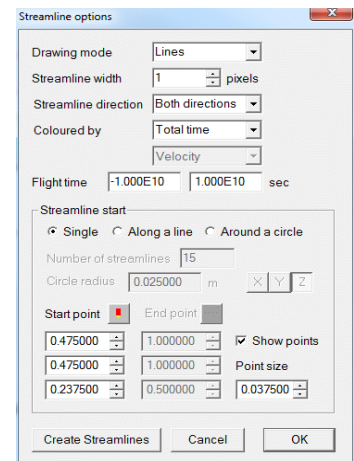
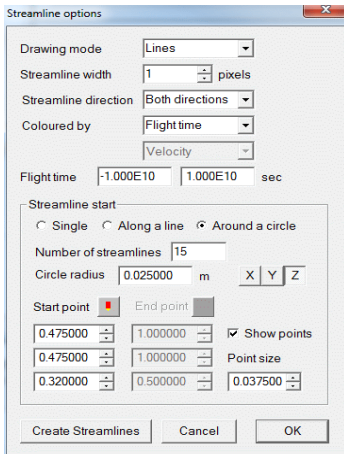Use of environment variables in filenames in PIL WRITE and EXEC commands is allowed.

Handling of quote marks in PIL character strings has been corrected.

Viewer

Streamlines

Dialog for setting streamline start positions has been updated:

The first image has too small a tolerance value. In the second image tolerance has been increased and the expected contour is seen.

Macro

File names longer than 20 characters can be used as input/output when extracting multiple data points.

Offset allowed for extraction location when extracting multiple data points. Extraction point coordinates can thus be relative to, say, a building origin, and the offset will convert to absolute coordinates relative to origin.

Correction to reading the number of colours to be used from a macro.

General

Plot surface objects can rotate freely.

Earth

ROTOR Object

ROTOR objects can overlap and/or touch. Previously they had to be separated by at least one cell in Y & Z.

ROTOR transient restart amended to ensure correct initial values are used.

Corrections made to diffusion term at ROTOR boundaries to prevent occasional failure with NaN.

General Improvements and Corrections

Corrections to shock corrections.

Transient term corrected on first step of restart to avoid huge time fluxes.

Capture source value from FIXVAL cells so that a sensible source value can be reported in the 'Nett Sources' section of RESULT.

The images show new dialogs for starting streamlines at a point along a line or around a circle. Everything required to specify the start location is on one dialog. Each time 'Create Streamlines' is clicked, a new set is created. On clicking 'OK' to close the dialog, the existing 'Streamline Management Dialog' showing all current streamlines is displayed.

SPARSOL

Small corrections to SPARSOL:

Add a search for minimum / maximum values in the cut-link section of the phi(da) file.
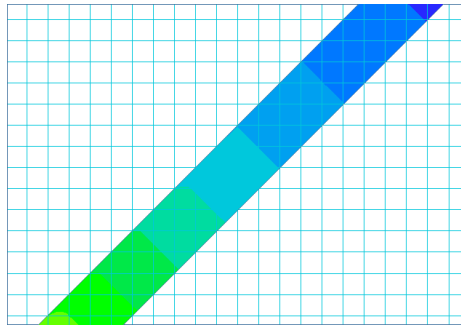
Don't try to read a pbcl.dat if one is present when SPARSOL is active.

PARSOL

Tolerance used in PARSOL contour plotting is now a CHAM.INI item.

`Graphics]PlotTol = 1.0E-7`

In some cases when facets pass exactly through cell corners, Viewer can get confused as to which side of the facet is solid and which fluid:

Implementation of fixed flux source in a PARSOL cut cell corrected.

Nett sources summary printed if there are more than 10 patches.

Sub-grid-size sources will still be applied (eg tiny heat source) in the cell(s) they are in, but blockage effects are ignored.

Additional InForm print frequency control NINFOR. INFROUT file will be written every NINFOR sweeps as well as at normal print sweeps.

Linear equation solver 1-D block correction sequence amended when solving H1, T3 or EPOT in conjugate situation if sources (and/or fixed values) are in solid cells.

In first and last cell adjacent to blockage only half-cell source is added; divide SHRX/Y/Z by 2 to ensure sum of SHRX*cell area will match nett source and friction force printed to result.

Prevention of Linux crash when saving final gxmoni plot image.

In SPARSOL cell-to-object linkage if object is above cut link, first cell is that with cell centre above intersection. If object is below, last cell is that with cell centre below intersection. This ignores cells with centres outside facets, as PARSOL did.

Same SPARSOL tolerance used for sequential and parallel to ensure same detection. In parallel the relative tolerance was based on local, not global, domain size.
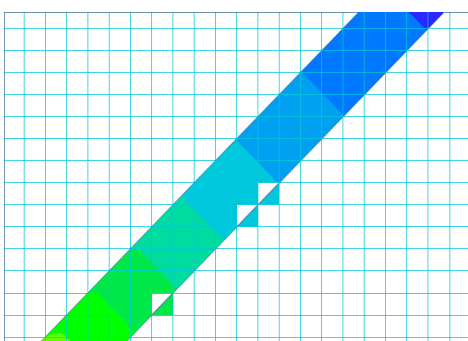
Allow GCV to store U/V/W1SL, U/V/W1AD, DP1X/Y/Z. U1SL are the post-solver UC1s, U1AD is the correction to U1.

Allow GCV to store PCOR.

Prevent a split in X direction for parallel if XCYCLE is used.

Issue error stop for ASM in parallel.

SUN object allows flow boundaries and selected blockage/plate objects to be transparent.

Optional new termination criterion: set ENDIT(phi) negative and run will stop when phi maximum absolute correction is below abs(ENDIT(phi)). For example, if ENDIT(P1)=-1, run will stop when the biggest absolute pressure correction is less than 1.0.

Print absolute maximum corrections (and their locations) to RESULT when printing residuals.

InForm CORR() operator corrected for variables solved whole-field to return correct values for slab-wise variables.

Flair

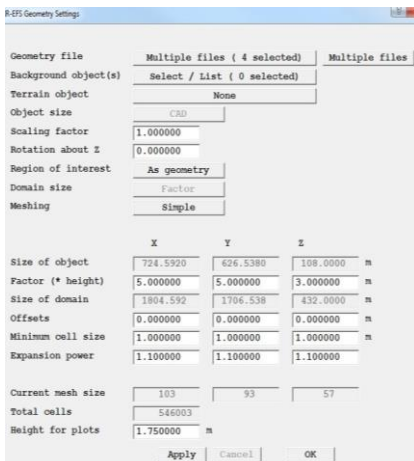Flair can show Pedestrian Wind Comfort and Averaged sector velocity, also in FLAIR-EFS.

Corrections to PMV calculation to ensure agreement with ISO 7730.

Flair-EFS

FLAIR-EFS is a reduced-cost subset of FLAIR for users concerned only with external environmental conditions surrounding buildings and other structures. Object types concerned with internal flows (ie jet fans, diffusers, spray heads) or fire modelling have been removed.

The normal mesh generator is replaced by a system based on size of building(s) being modelled, which are always placed centrally in domain. Domain size is based on multiples of maximum building height.

Users set a minimum cell size. FLAIR-EFS meshing dialog is shown below:

FLAIR-EFS can show Pedestrian Comfort information including:

- Wind Amplification Factors.
- Probability of the wind speed to exceed a set threshold value.
- NEN8100 Dutch standard for Wind Comfort and Wind Danger in the Built Environment.

Given a table of wind direction and wind speed probabilities for each direction, it is possible to create an average wind speed by running for each wind direction in the table, and then summing absolute velocities multiplied by sector probabilities. The PHISUM utility can be used to sum output files from individual runs.

ESTER
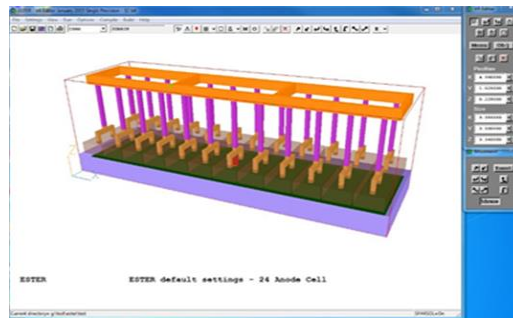
See typical ESTER setup above.

The ESTER user interface has been extensively upgraded to include:

- Cathode and collector bar configurations below metal pad
- Anode stubs, rods and bus bars above electrolyte layer
- Additional post-processing output, including:
- Interface height surface
- Interface height map
- Voltage & current monitoring files
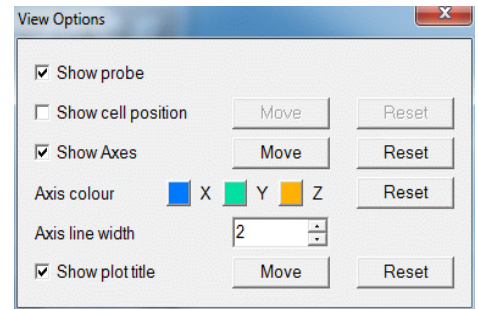- Macros to create vectors from currents, magnetic fields and Lorentz Forces

CVD

Allows blank and comment lines (starting !) in THERMDAT.

Allows blank lines in CHEMIDAT (which allowed ! comments).

Increases maximum number of reactions to 100.

General, VRE & VRV

Colour and line width for domain and object axes can be set (and saved to Viewer macro):


Toolbar and handset icons have been updated for better legibility when using Windows high contrast theme; icons now look like this:


New PHIDIFF utility to subtract two phi(da) files (run from 'Run/ Utilities'). Useful for identifying differences between two solutions.

New PHISUM utility to sum several phi(da) files (run from 'Run/ Utilities'). Useful to sum average velocity over several wind directions.

Tutorial and documentation files can be specified in phoesav.cfg and then appear in the 'Help' menu:
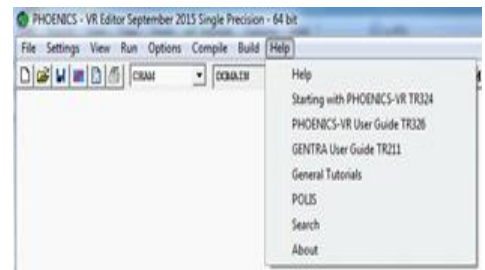
Dialogs can be displayed in other languages with language file present (sample Chinese menu below):
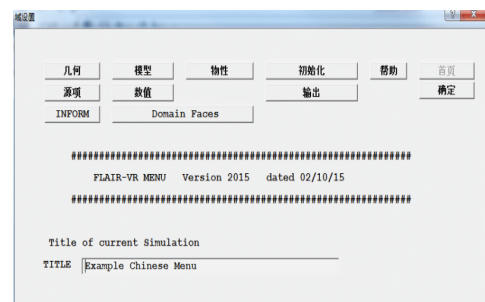
Only licensed SPPs can start.

PHOENICS modelling of a moving surf generator using MOFOR

Dr R. P. Hornby
(bob@hornby007.wanadoo.co.uk)

On August 1St 2015, after a Welsh Government contribution of £4.15million to the total cost of £15million, the world's first inland surf lagoon was opened at Dolgarrog in Wales. The facility is expected to attract 75,000 people every year and host international surfing events featuring the world's best surfers.

The surf park employs a unique wave generating technique – a moving wave foil that produces powerful and consistent waves of varying heights up to 2m in a 300m long by 110m wide lagoon.

The 'snow plough' shaped underwater wave foil is positioned under a pier running the length of the lake (figure 1) and is pulled along a central underwater track, producing a barrelling wave either side of the foil giving about 20s of surfing time (figure 2).

Steel mesh isolates the foil from nearby surfers for safety reasons and a porous grid sheeting is used to damp waves at the boundary.

Figure 1. Detail of the wave generating mechanism.

The details of the wave foil design and lagoon contours are commercially sensitive and have been developed over a number of years**.** However, it is shown here that PHOENICS has the appropriate methodology to contribute to such studies, in particular, having a free surface model (SEM the Scalar Equation Method) and a moving body facility (MOFOR).

Figure 2. Surfing waves generated either side of the surf lagoon pier.

PHOENICS modelling

For the purposes of the modelling exercise a 50m long by 30m wide half section of a surf lagoon is simulated. A maximum water depth of 2m is taken, shoaling to 1m depth at the far end of the simulation domain. A 4m air gap is used over the water.

The whole domain is discretised using a Cartesian grid with 55 cells in the x (lateral), 130 in the y (longitudinal) and 30 in the z (vertical) directions. A finer grid would need to be used to capture the full details of the wave breaking process but the current grid is considered sufficient to assess viability of this modelling approach.

Because the modelling will use a moving body no clear advantage is obtained by employing a non-uniform grid (at least in the

direction of motion of the body), so for simplicity the grids used here are taken to be uniform in each coordinate direction.

The flow is assumed laminar with smooth wall friction applied on the bottom boundary. The top boundary is taken as a constant pressure boundary.

The wave foil is assumed to be wedge shaped with a vertical surface 4m high and 3m wide generating the wave. A sloping back on the wave foil ensures that water cascades over the foil in a controlled manner (figure 3). The speed of the wave foil is taken just greater than the maximum wave speed to ensure an attached, nearly transverse wave. Wave foil motion is enabled by the following illustrative statements in the Q1 file. The speed used in the simulation is set constant and equal to GV along the y axis, but any variation with time (TIM) can be used.

```
PATCH(MOFOR,VOLUME,1,1,1,1,1,1,1,LST
EP)
SPEDAT(SET,MOFOR,MOFFILE,C,NOTSET)
(MOVOBJ OF B1 IS POS
(0.0,GV*TIM,0,0,0,0))
```

where B1 is the wedge object (the wave foil referred to above) set into the domain using the VR editor.
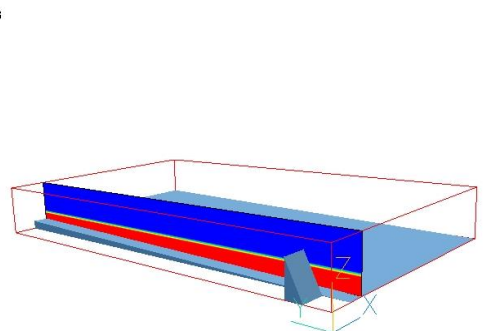
Figure 3. The wedge object representing the wave foil shown positioned in the simulation domain (with sloping bottom).

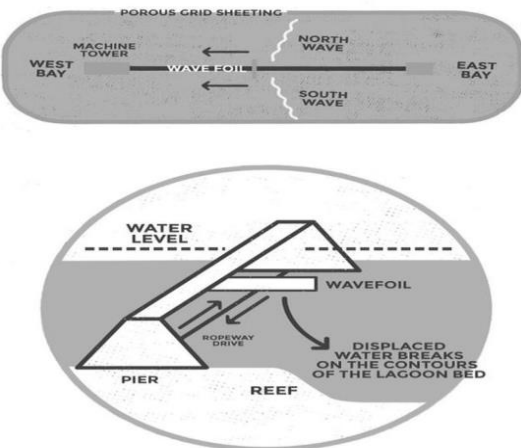The SEM is used which tracks the water/air interface and allows overturning. This method has been used in a separate study on water wave propagation with good results (Ref 1).

As this is an explicit method the time step taken needs to ensure that the Courant Number is less than 1.0.

In the present calculations the time step is taken as 0.02s and the simulation is run for 10s.

This time is sufficient to initially assess the wave generation capability of the modelling which can then be easily extended (e.g. using more grid cells) to capture additional detail of the wave generation processes.

Results

Figure 4 shows that after 1.8s a wave in excess of 2m height is formed at the foil. The SEM also nicely accommodates the flow over the foil and down the back slope.
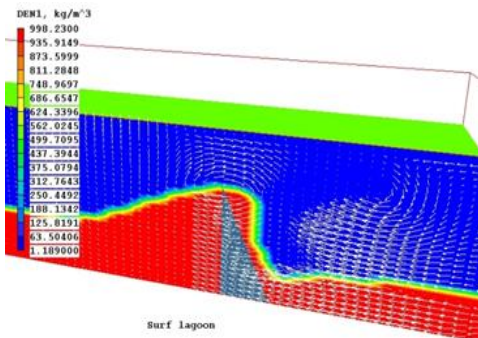

Figure 4. Density distribution in a vertical plane at the wave foil showing the formation of a 2m high wave after 1.8s..

Figure 5 shows the wave heights (GWHT, units metres) in the surf lagoon after 6.9s. The wave is seen to fan out from the foil rather like the waves in figure 2 with wave height decreasing with distance from the wave foil.

However, no attempt has been made here to refine parameters to encourage a wave breaking scenario as the current grid is not sufficiently refined to capture this process.

What is in evidence is a smoothed version of any wave breaking process.
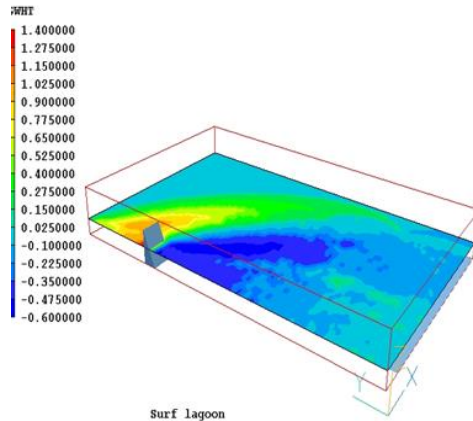
Figure 5. Wave heights (m) after 6.9s.

Conclusions

PHOENICS MOVOBJ methodology combined with the SEM provide flexible tools to examine surfing wave generation in leisure facilities.

The SEM, although explicit, is well suited to surfing event times of order 20s although a refined grid is required to capture the sharpness of wave profiles near to breaking.

The MOVOBJ method allows the shape and speed of the wave foil to be easily modified in order to examine the formation of surfing waves but necessarily implies use of a uniform grid in the direction of motion (a costly requirement in terms of computer time).

References

1. PHOENICS modelling of 3-D flow over a surf reef and comparison with experiment. PHOENICS Newsletter, Winter 2012.

Drainage of Water Droplets in a Bounded Paraffin Oil Continuous Phase: Role of temperature, size and boundary walls

Jalil Ouazzani, ArcoFluid, France
Adil Lekhlifi* and Mickael Antoni*
** University of Marseille, France*

The full article presents numerical simulations of the hydrodynamics generated by droplets evolving in a continuous paraffin oil phase and submitted to gravity.

The model is two-dimensional, includes free boundaries, capillarity, and is simulated with PHOENICS using ground coding.

Droplets are confined in a two-dimensional simulation domain of 1 cm side. The influence of their size and temperature in the overall drainage dynamics is described.

The initial condition is discussed for 1 mm radius droplets to describe the effect of boundary walls.

Changes in the circulation flows are evidenced, different hydrodynamic regimes identified and their characteristic times discussed.
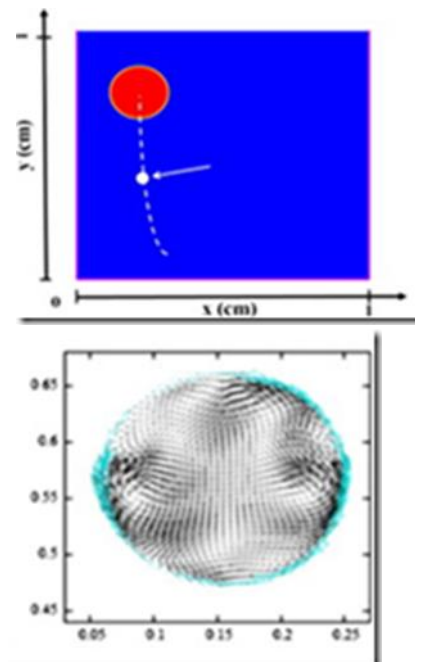
Projects Using PHOENICS Undertaken by Students from Moscow Power Engineering Institute

Vsevolov Volgin, MPEI, Russia
**Yaroslav Volgin,** MPEI, Russia

We are second year graduate students from Moscow Power Engineering Institute (MPEI) who have used PHOENICS in our studies to model different processes in solid objects, liquids and gases. In August 2015 we visited CHAM to gain experience using PHOENICS by working with its developers and to learn about new functions and possibilities. Our work is outlined below.

Vsevolod worked on thermal processes in thermoelectric cooled units, called TEC's. By using one of the new PHOENICS SimScenes he created an interface allowing Users to set necessary parameters: size, configuration, material properties.

Temperature field distributions in the unit necessary to determine the efficiency of energy transformation, electricity power and energy losses in the TEC design, are calculated.
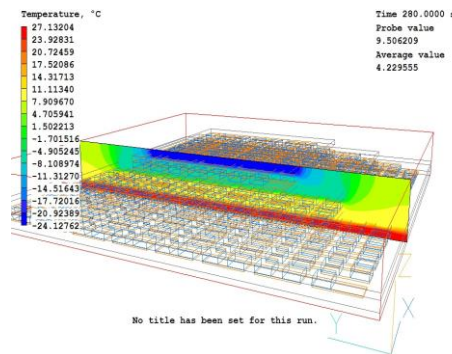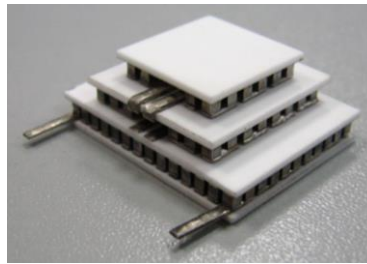

Jaroslav's project was to simulate the nanopowder synthesis process using a gas phase synthesis method. It contains two stages: evaporation of material and nucleation of vapour with further conversion into particles. At CHAM Jaroslav worked on the first stage. It is important to understand how vapour distribution develops in the domain and how environmental conditions at the initial time affect it. PHOENICS was used to calculate evaporation, heat and mass transfer processes, to obtain the distribution of vapour concentration and temperature in the space. This information will be needed for the second stage.

Atomizer with evaporation.
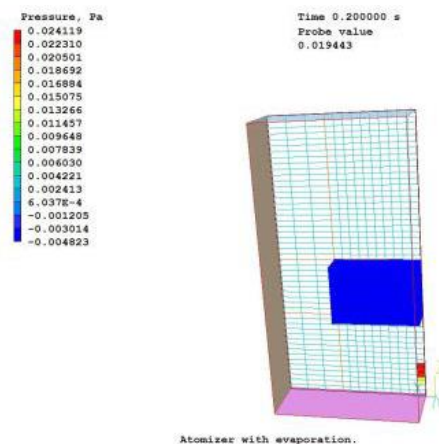
During the visit we learned much about PHOENICS and its functions. In particular, Vsevolod learned about using "objects" and the principles of setting boundary conditions. Jaroslav familiarized

himself with GENTRA - which calculates individual particle trajectories and would be useful for developing the nucleation model in the second stage of his project.
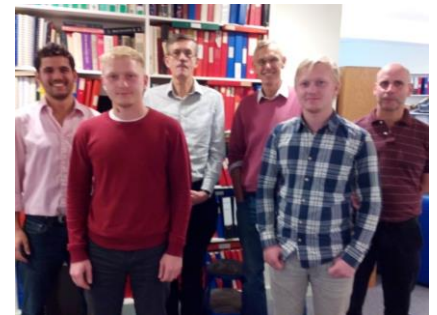
Vsevolod & Jaroslav Volgin with CHAM Staff Summer 2015.

We would like to thank CHAM employees for their warm attitude to us. It was an honour to talk with Professor Spalding - a prominent scientist, highly respected in Russia. Special thanks go to Andrew who supervised and helped us. We really enjoyed our time in England and thank CHAM for new experiences.

Critical Anomaly Effects in Supercritical fluid Flows in Capillaries

Bernard Zappoli, CNES, France
Jalil Ouazzani, ArcoFluid, France
**Carole Lecoutre*, Samuel Marre*,
Sandy Morais*, Romain** Guillaument*
and Yves Garrabos*,
*ICMCB, CNRS, France

Structure and basic thermal and dynamic properties of steady state, near-critical fluid flows in capillaries are explored by numerical analysis of Navier-Stokes equations written for a Newtonian, heat conducting and viscous van der Waals fluid. We focus on where and how critical transport coefficient anomalies dominate the fluid flow in a capillary duct of constant section.

PHOENICS software from CHAM was used to simulate these high Mach number critical flows ($0.4 < Ma < 1$) in 2D axisymmetric configurations and for several gas, CO_2 and H_2.

11th Int. Conference: Heat Transfer, Fluid Mechanics & Thermodynamics (HEFAT 2015), Kruger National Park, South Africa, 20-23 July.
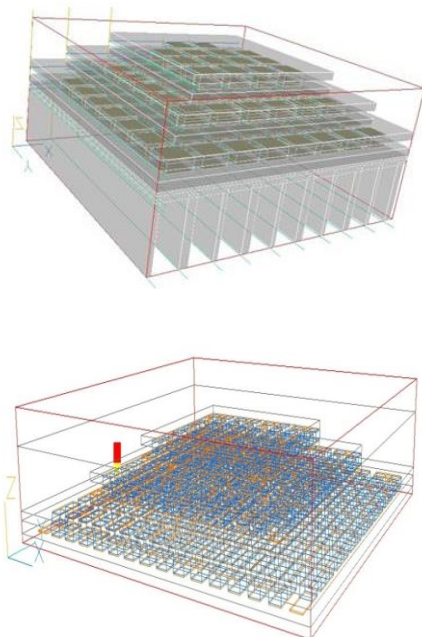
The Effect of Student Distribution on the Natural Airflow Pattern of a Classroom

C. Morales Maya1, M. Ordenes Mizgier2
1,2 Federal University of Santa Catarina, Florianópolis -SC, Brazil.

Abstract

People´s thermal sensation inside a room, and the awareness of whether they feel comfortable or not in it, are highly influenced by ventilation. Every type of environment has different requirements to guarantee thermal comfort. These requirements are mainly oriented by the activity which inhabitants perform in the specific environment, and the amount of people that use it.

School environments, such as classrooms, need great care in this respect. Not only do they host one of the most important activities in society, education, but they shelter a large number of people performing educational activities simultaneously during extended periods of time. Therefore, school environments are strictly regulated in this regard.

Brazilian standards for educational buildings give specific requirements about the amount of open façade necessary in classrooms to fulfil this purpose, and other complementary information. They also enunciate the importance of the flexibility of these spaces, in order to contain activities that respond to different educational practices and philosophies.

Basic requirements regarding the design of school environments are listed in a series of documents called Technical Notebooks. "Elementary School Educational Spaces. Grants for projects in development, and adequacy of school buildings, technical section 4 volume 2" contains the main design parameters of classrooms, including mandatory openings for natural ventilation.
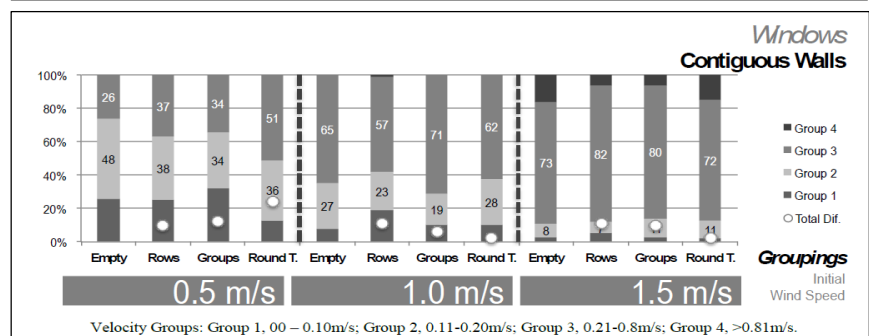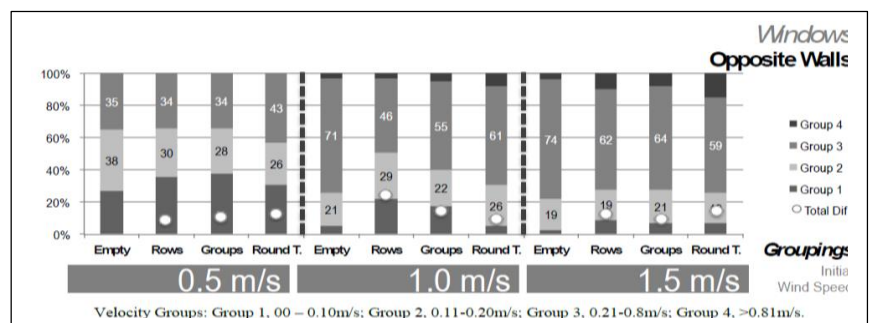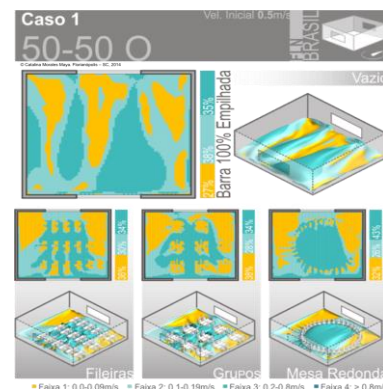
Inside the space, the human body can modify the airflow pattern acting as a physical obstacle and a heat source. Conditions in these spaces, which are used by a considerable number of people, are dependent on the way users group during their normal activities, which change constantly.

Taking into consideration the observations above, the aim of this study is to evaluate behavior of air flow in a hypothetical classroom, while the space is empty and when occupied by 36 students grouped in different ways. The classroom was designed according to the parameters listed in the Technical Regulation of the Brazilian Education Ministry, and took as a reference, summer weather conditions of Florianópolis-SC.
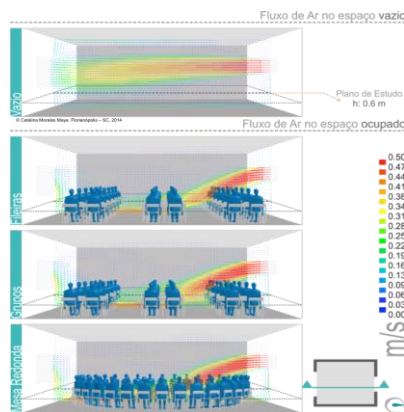
This research used PHOENICS, a Computational Fluid Dynamics (CFD) code, with a specific module (FLAIR) for architectural purposes. The total ventilation area required, adopted from the standard, is divided into two openings located in two space façades thus obtaining different classroom types. Velocity values were obtained for each point of the classroom, at height 0.6m. Each evaluated plane point was classified in one of 4 velocity groups, low, medium-low, medium or upper limit speed.

Data analysis showed that, with an average of 14% of the evaluation plane area that shifted from one group to another when comparing empty and occupied classrooms, occupation had a great influence on modification of initial distribution of air velocity at height studied, without necessarily reducing wind speed.

Velocity Groups: Group 1, 00 – 0.10m/s; Group 2, 0.11-0.20m/s; Group 3, 0.21-0.8m/s; Group 4, >0.81m/s.

Velocity Groups: Group 1, 00 – 0.10m/s; Group 2, 0.11-0.20m/s; Group 3, 0.21-0.8m/s; Group 4, >0.81m/s.

Agent News

ACADS-BSC *Agent for Australia*

ACADS will exhibit at ARBS 2016.

Safe Solutions *Agent for Brazil*

A workshop was held, on May 25, at UFSC (Florianópolis) to discuss PHOENICS. It was attended by teachers and students from Engineering and Architecture. Information was provided on uses of PHOENICS and there were opportunities to discuss activities being undertaken by those present. For further information contact fabio.fundo@safesolutions.com.br

Shanghai-Feiyi *Agent for China*

Shanghai Feiyi held a Green Building Training Course in Shenzhen on September 15-16 2015.

The first photo shows Shanghai Feiyi's engineer giving a demonstration of a single-room natural ventilation simulation using PHOENICS.

The second photo is of the audience being shown an example of a large area Air Ventilation Assessment (AVA) study using PHOENICS.

CHAM News

CHAM is pleased to welcome two new Engineers to our team. They are Matthew Evans (left) and Tom Horsefield. (right) below; we look forward to working with them.

We said farewell to one of our engineers at the end of September when Marta Camps Santasmasas left. We wish her well.

Advances in Computational Heat Transfer: 6th Symposium

The International Centre for Heat & Mass Transfer Symposium, (May 25-29 2015, Rutgers University, USA) was dedicated to *Professor Suhas Patankar* and honoured *Professor Graham de Vahl Davis*.

Brian Spalding and Suhas Patankar at the podium at CHT-15

Brian gave a keynote lecture on *Water Cooled Condensers* (see http://www.cham.co.uk/DOCS/papers/Rutgers_Lecture_May_2015.pdf)

The scientific programme comprised invited lectures by distinguished researchers, contributed papers, and panel sessions. An Open Forum meant authors could present new work, or work in progress

Other Keynote speakers included Suhas Patankar, Bengt Sunden, W.Q. Tao, Q. Wang, Philip Allen, Wojciech Lipinski, & S.P. Vanka.

Turbulence, Heat and Mass Transfer: THMT 8th Symposium

Brian was invited to give a *Special Lecture* at this Symposium in Sarajevo, Bosnia Herzegovina, September 15-18 2015. His subject was *A Turbulence Model for the Fully-Stirred Reactor*

Abstract: The problem discussed is how to calculate the shapes of the probability- density functions which characterise the turbulent fluctuations within a 'fully-stirred reactor'. This is an idealisation of the paddle-stirred reactor used in chemical-process industry; and of the combustion chamber of a gas-turbine engine. Its defining characteristic is having the same time-average state of the fluid mixture at every location within it.

Keynotes included: B Launder, D Markovich, M Oshima, Y Suzuki, M Pourkashanian, and C Tropea.