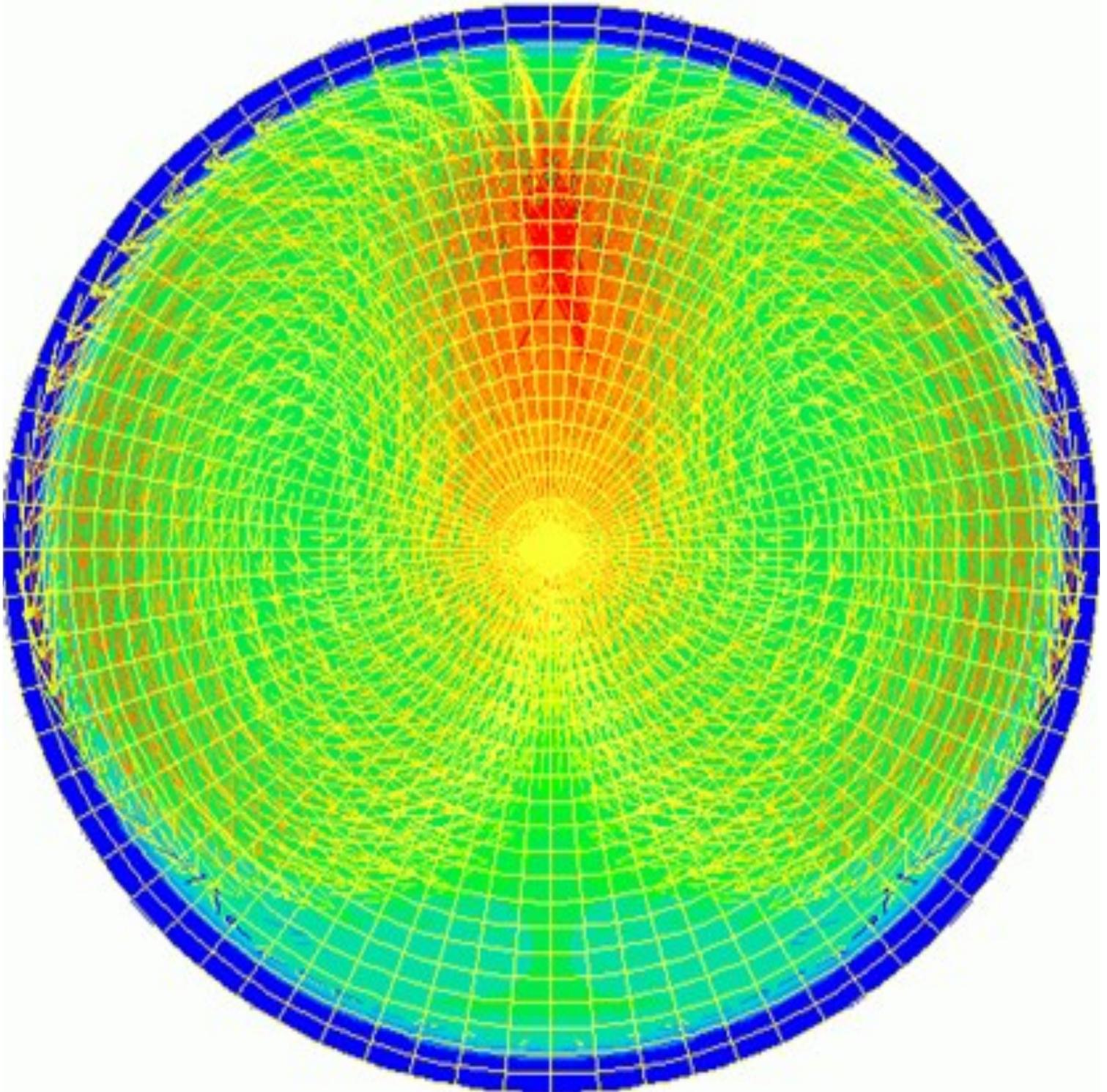


# ***PHOENICS NEWS***



***PHOENICS – Empowering Engineers***

**Spring 2016**



## ***Simulation of a New CSO Chamber Design for the Water industry: CHAM Consulting Team Case Study***

A combined sewer handles both sewage and rainwater runoff from streets. During storms, sewer flow can be many times the dry-weather foul-flow rate which can cause flooding from the sewer network and problems at treatment plants. Overflow arrangements are provided to divert excess sewer flows into a river or other receiving waters during storms. These relief structures are known as Combined Sewer Overflows (CSOs).

One problem with CSOs is that sewage debris and litter of various kinds washed into the drains by a storm can foul the discharge screen or be discharged from the CSO into the natural environment. Figure 1 shows a CSO spilling litter to receiving water due to a blocked screen.

WRc plc ([www.wrcplc.co.uk](http://www.wrcplc.co.uk)) has devised an innovative, non-powered CSO which is designed to reduce this form of pollution. A CSO typically works by having a chamber in which the water level rises in storm conditions until it reaches a high-level weir which provides the overflow

In WRc's patented design, the inflow into the CSO first passes through a primary (stilling) chamber where separation of litter (solid matter of low density) occurs. The litter passes over a low level weir in the primary chamber into a pipe which bypasses the spill chamber and rejoins the sewer downstream. The pass forward flow, free of litter, passes into the downstream (spill) chamber where, with the increase in surcharge of the system, discharge from the CSO via the screens. This arrangement prevents litter from fouling the discharge screen or being released into the environment due to overtopping of blocked screens during storm responses.

Asset International built a prototype CSO chamber to demonstrate the design principle that was exhibited at WRc's 2015 Innovation Day (Figure 2).

CHAM's consulting team was invited by WRc to perform a CFD analysis of the filling and emptying process for the new CSO design, in order to provide proof of concept results for its operation. CHAM's objective was to demonstrate that the CSO would successfully separate the bulk of the accumulated litter in the CSO primary chamber during filling and emptying.

Earlier simulations using CHAM's PHOENICS software produced promising results concerning design and testing of a steady-state CFD model of the innovative CSO during peak flow. The CFD model complied well with design parameters and successfully demonstrated separation of materials in the chamber at specified flow rates when the primary chamber low-level weir was flooded. However, the model had not been applied to transient (time-dependent) filling and emptying process encountered in a storm surge.



**Figure 1: CSO spilling litter due to blocked screen**



**Figure 2: Downstream end of pre-chamber**



## **CHAM News**

CHAM will be exhibiting **PHOENICS** at the forthcoming **WRc Innovation Day April 27 2016 @ WRc, Swindon.**

[http://www.wrcplc.co.uk/Data/Sites/1/media/pdfs/events/Agenda%20V3%20Web%20Version%20\(2\).pdf](http://www.wrcplc.co.uk/Data/Sites/1/media/pdfs/events/Agenda%20V3%20Web%20Version%20(2).pdf)



The new study objective, again sponsored by WRc, was to extend the earlier CFD model and capture features including modelling filling and emptying to demonstrate separation efficiency of the primary chamber and bypass pipe in carrying away unwanted litter material at various water levels; viz, when the weir in the primary chamber was (a) not operating, (b) spilling, and (c) flooded. The double chamber CSO CFD model geometry was constructed from drawings supplied by WRc.

The new CFD model was viewed as an aid to the pre-construction design process; it was able to predict how varying parameters such as dimensions of the overflow chamber and weir height would affect the efficiency of litter separation.

The objective was to provide WRc with a generic modelling tool which could be used to analyse the design and, ultimately, to enable WRc to demonstrate to clients that the new CSO chamber design will reduce both litter discharge to the environment and blinding of the CSO screen.

The most challenging aspect of the project was to determine a robust technical approach to handling the free water surface in the model. In the context of the CSO, it was clearly important that the rise and fall of the surface level could be handled.

The methodology chosen to represent the three phases present (water, litter, air) was the PHOENICS Algebraic Slip Model (ASM). based on the modelling assumption that there is a single continuous phase containing other dispersed components whose motion (slip velocity) relative to the continuous phase is given by an algebraic relationship involving local hydrodynamic variables.

Local volume fractions for each component (including the continuous phase) are determined from individual conservation equations, with the constraint that component fluxes across each cell face should be compatible with the corresponding mixture flux. This method allows for multiple phases to be computed and provides a way of describing the free-surface formed between the water and the air.

The time-varying inflow rate of water and litter into the domain, representative of a storm event, was described by a curve provided by WRc and input straightforwardly into PHOENICS using the "INFORM" (INput of FORMulae).

The proportion of litter introduced via the inlet varied in accordance with the inflow rate, ranging from an initial base rate of 5% by mass, escalating to 20% at peak inflow rate, reducing thereafter. This curve was specified as a fill/empty cycle over the course of an hour.

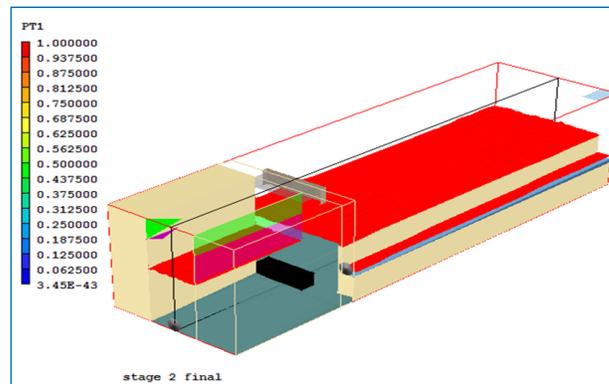


Figure 3 gives a snapshot of the model, showing water surface levels after 1400s (23 minutes); the green rectangle at the top on the left represents the open space above the discharge screen (represented by the purple rectangle). The small red rectangle below it is the water surface in the spill chamber and the large red area on the right is the water level at the bypass weir in the primary-chamber, which can be seen lapping the weir. The thin red area on the right is the flow in the bypass overflow channel beneath the weir.

The results were provided in the form of a table showing the proportions of the water and litter flowing into the CSO chamber that are (i) bypassed to the downstream sewer, and (ii) passed forward via the continuation pipe to the spill chamber; and also, the proportions of flow and litter discharged to the receiving water under varying circumstances.

The promising results obtained from the CFD models are being used by WRc in the design of a CSO that is to be trialled live shortly to confirm the CSO operates to expectations.

CHAM will be exhibiting PHOENICS at the forthcoming **WRc Innovation Day, April 27 2016 @ WRc, Swindon.**

[http://www.wrcplc.co.uk/Data/Sites/1/media/pdfs/events/Agenda%20V3%20Web%20Version%20\(2\).pdf](http://www.wrcplc.co.uk/Data/Sites/1/media/pdfs/events/Agenda%20V3%20Web%20Version%20(2).pdf)

## Modelling Crowd Behaviour Using PHOENICS

by Bob Hornby

### Introduction

Being able to predict how large crowds behave is important in ensuring the safety of people taking part in religious festivals, sporting events, music festivals, national celebrations, demonstrations or in movements of people in airports, stations or passenger ships etc.

As an example, a number of Hajj-related crowd crush disasters have occurred in the past. The Hajj is an annual pilgrimage in Mecca prescribed as a duty for Muslims to undertake at least once in their lifetime if they can afford to do so physically and financially. Over 2000 deaths were recently reported in a stampede on 24 September 2015 as pilgrims were en route to the Jamarat Bridge, a pedestrian bridge in Mina, Saudi Arabia, near Mecca (figure 1).



Figure 1: Crowds of pilgrims making their way to the Jamarat Bridge (2011)

There are two basic ways of modelling crowd behaviour (Ref 1). In the first the crowd is considered to be made up of individual entities which obey certain rules and the aggregate of these determine the crowd behaviour. This is akin to a Monte-Carlo approach and becomes expensive when large numbers of entities are involved. The second approach is to model the crowd as a 2-D continuum obeying mass and momentum conservation. The latter approach is particularly suitable for Computational Fluid Dynamics and it is interesting to see how this could be explored using PHOENICS.

### PHOENICS modelling

As a modelling exercise, crowd behaviour is considered in a 50m by 50m area with up to two exit points (more can be added if required). Initially a high crowd density is considered at the centre of the area and the evolving density distribution and crowd flow is investigated via a single exit. More complicated situations are then considered involving more than one exit location and the effects of a blockage. The standard PHOENICS mass and momentum equations are used with the pressure related to the density (compressible flow) and source terms included in the momentum equations to model crowd motive and directional effects. The details are given below.

The pressure ( $p$ ), density ( $\rho$ ) relationship is simply taken as,

$$p = C\rho$$

where  $C$  is a constant of  $O(1)$  (Ref 1). This implies a signalling speed (when pressure provides the dominant force) equal to  $C^{0.5}$ .

The source terms in the momentum equations account for motive force and 'drag' (assumed proportional to the velocity) on individuals. It is assumed that these forces are directed towards the nearest exit or nearest visible exit. When the density is low the motive force is expected to dominate but when the density is large the pressure force becomes dominant so the motive force is linearly interpolated with density. The volume source term is

$$\frac{\rho(f - \beta V)}{m}$$



where the vector  $f$  is the interpolated motive force and  $V$  the velocity vector, both directed to the nearest exit or nearest visible exit.  $\beta$  is a constant and  $m$  the average person mass. The maximum velocity,  $V_{max}$ , in regions of lowest density is given by

$$V_{max} = \frac{|f|}{\beta}$$

with time constant  $m/\beta$ .

The domain is discretised using a Cartesian grid with 50 cells in the  $x$  and  $y$  coordinate directions. The KOREN scheme is used with a time step satisfying the Courant condition. A localised

$$\rho = \rho_{min} + (\rho_{max} - \rho_{min}) \exp(-\alpha((x - x_c)^2 + (y - y_c)^2))$$

crowd density at  $(x_c, y_c)$  is taken in the form

where  $\rho_{max}$  is the maximum (localised) density,  $\rho_{min}$  the minimum (background) density and  $\alpha$  a constant determining the size of the crowd.

Boundary surfaces are taken as free slip – shear effects, which may be significant when the density is large, are not considered in the current form of the model.

At the exit points the pressure is simply taken as fixed (at  $0.1Pa$ ), though there is obvious scope for more realistic exit conditions (which would include the shape of exits and possible choking).  $C$  is taken as 1,  $m$  as  $100kg$ ,  $V_{max}$  as  $2.5m/s$ , the motive force as  $100N$  and  $\rho_{max}$  as  $200kg/m^3$  (i.e. 4 people/ $m^2$  in a 2m high domain).  $\rho_{min}$  is taken as  $0.1kg/m^3$ . These are considered as reasonable rather than definitive values.

## Results

The first case considered has a localised crowd distribution at the centre of the domain with an exit in the top right hand corner at  $(a,b)$  where  $a$  and  $b$  are the domain sizes in the  $x$  and  $y$  directions. It is assumed that some disturbance takes place which requires an evacuation of people.

The results are shown in figure 2. The crowd movement is almost immediately directed towards the exit and continues so with time, symmetrically distributed about the diagonal. The velocity magnitude in the low density regions also corresponds closely to the maximum predicted velocity,  $V_{max}$ .

Figure 3 shows the same situation but with two localised crowd density distributions. This shows a merging of the two localised crowds towards the exit, in this case with a larger ‘flow’ below the diagonal.

Figure 4, on the following page, shows a more interesting case with two localised crowd distributions and two exits, one at the top left of the domain and the other at the top right of the domain. Here the motive force of individuals is directed towards the nearest exit (as might be expected in practice) and this is clearly illustrated in the results which show a pronounced bias towards the right hand exit.

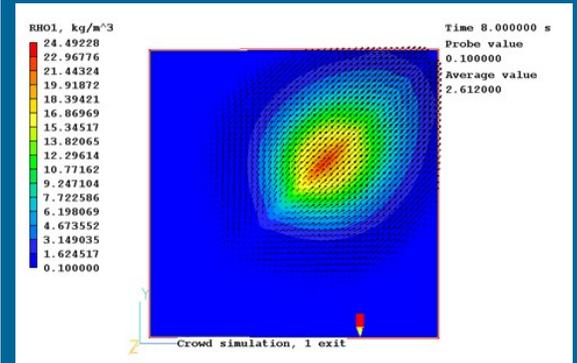
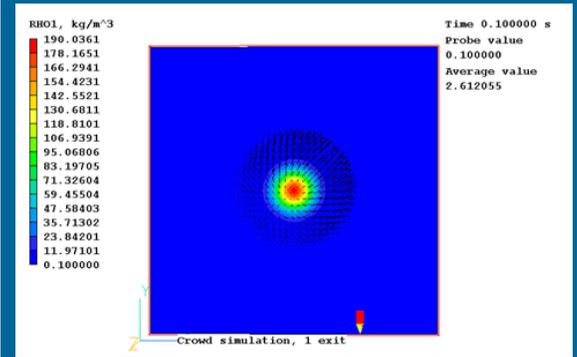


Figure 2. Crowd density and velocity vectors after 0.1s (top) and 8s (bottom). Exit is situated at the top right hand corner.

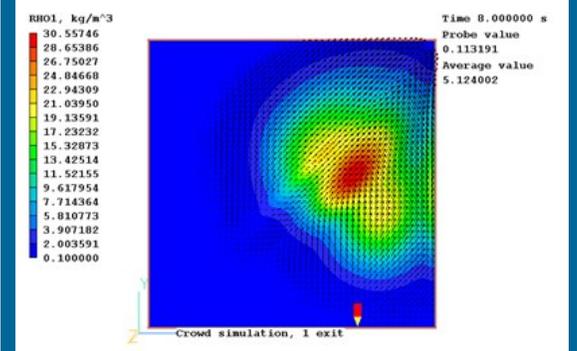
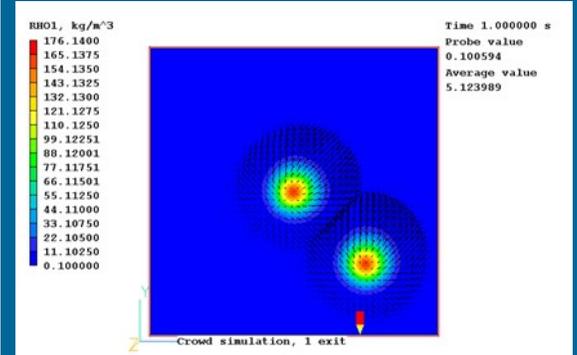


Figure 3. Crowd density and velocity vectors after 0.1s (top) and 8s (bottom). Exit is situated at the top right hand corner.

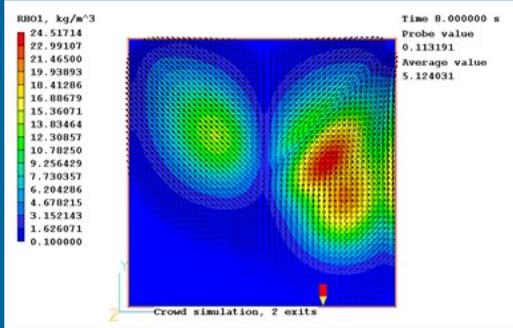
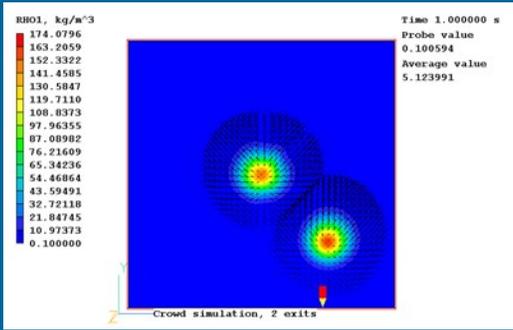


Figure 4. Crowd density and velocity vectors after 1 s (top) and 8 s (bottom). There are two exits at the top left and top right hand corners.

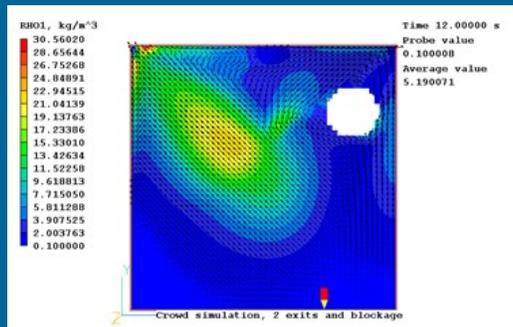
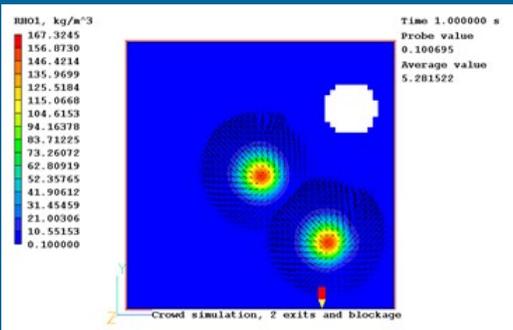
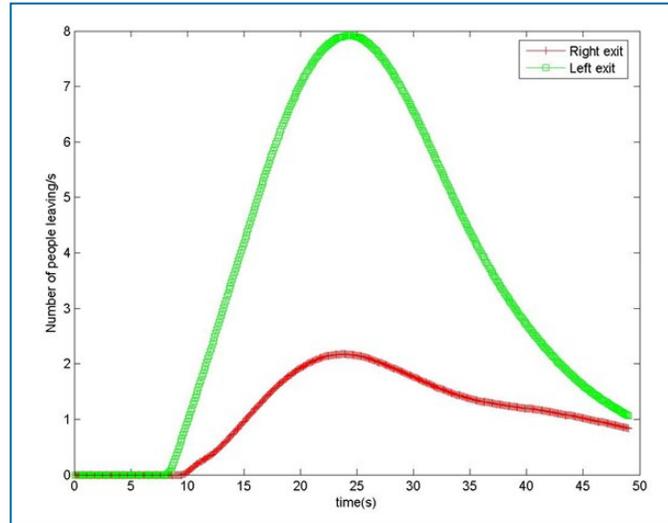


Figure 5. Crowd density and velocity vectors after 0.1 s (top) and 8 s (bottom). There are two exits at the top left and top right hand corners and a circular blockage near the top right exit.

Figure 5 shows results with two localised crowd distributions, two exits and a circular blockage. The blockage will impede flow but will also tend to obscure the top right exit.

This effect is included in the modelling which assumes that the crowd moves towards the *nearest line of sight exit*. This is clearly illustrated in the results with the majority of movement directed initially towards the left hand exit because the right hand exit is unsighted.

This is exemplified in figure 6 which shows the rate at which people move through each exit as a function of time. It takes a finite time for people to reach the exits, but the left hand exit



always shows the larger efflux which peaks after about 25s.

Figure 6. Number of people leaving the domain per second through each of the two exits when there is a circular blockage near the top right exit.

## Conclusions

It has been shown that with some fairly simple, physically motivated modelling additions PHOENICS can produce quite plausible results for crowd behaviour. The computer runs converge very well on each time step and being 2D they do not require significant computer resource. However, additional consideration needs to be given to the geometry of the exits which may limit exit rates.

Comparison with results from actual crowd movements and subsequent refinement of the modelling is required, leading to the capability to investigate more complex scenarios (as in figure 1) where unusual crowd behaviour can lead to loss of life.

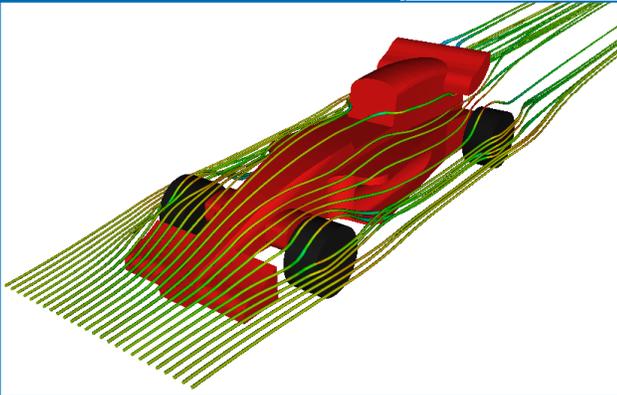
## References

Al-Nasur, S. and Kachroo, P., "A Microscopic-to-macroscopic crowd dynamic model" (2006). *Faculty Publications (ECE)*. Paper 93. [http://digitalcommons.library.unlv.edu/ece\\_fac\\_articles/93](http://digitalcommons.library.unlv.edu/ece_fac_articles/93)

Dr R. P. Hornby e-mail: [bob@hornby007.wanadoo.co.uk](mailto:bob@hornby007.wanadoo.co.uk)

# PHOENICS® to OpenFOAM® Interface [PH2OF]

Jason Cooke, CHAM.



PHOENICS F1 VWT case replicated in OpenFOAM and visualised using ParaView

## 1. Overview

CHAM has added to its PHOENICS CFD package a special “PH2OF” (PHOENICS-to-OpenFOAM) ‘translator’ so that users are able to utilise the CFD solvers of both codes. This document describes the functionality of the PH2OF translator.

OpenFOAM is a free, open source CFD software package developed by OpenCFD Limited for the ESI Group and distributed by the OpenFOAM Foundation. – [www.openfoam.com](http://www.openfoam.com).

The translator can be run using two different approaches:

- From within the PHOENICS VR-Editor environment.
- Using the command line and the provided batch script in the working directory, without using the VR-Editor environment.

## 2. How to use PH2OF

### 2.1 Within VR-Editor

The simulation can be set up as a New Case in the usual manner within the VR-Editor. Alternatively, pre-existing library cases can be loaded from the PHOENICS libraries.

For cases containing faceted objects (with faces not aligned with grid lines), it is important that ‘Additional individual STL output’ is activated in VR-Editor. Also, the probe monitor must be positioned within the fluid region of the grid.

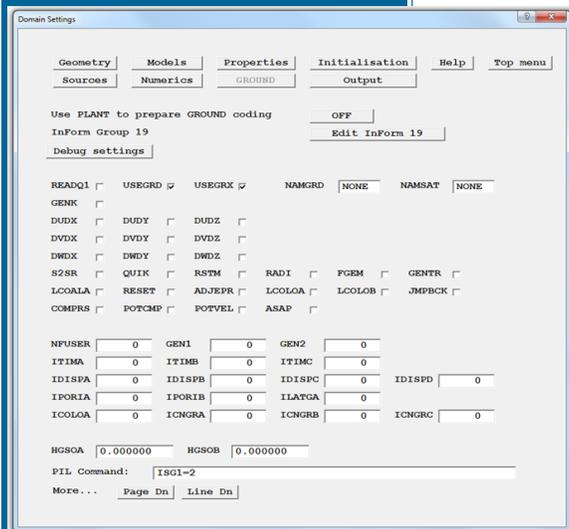
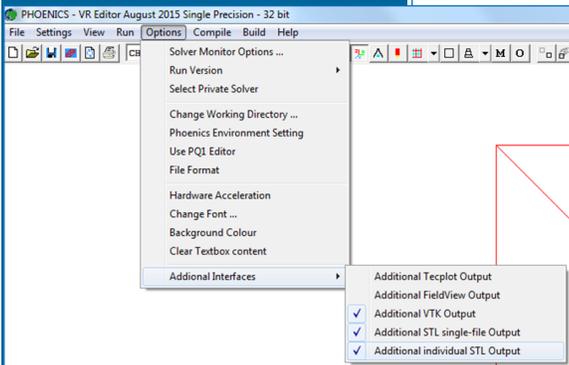
Optionally, ‘Additional STL single-file output’ can be selected to output all domain objects in a single STL file, whilst ‘Additional VTK output’ will output the results of a PHOENICS-EARTH simulations in VTK format. This enables PHOENICS-EARTH results to be viewed in ParaView®, alongside OpenFOAM results. ParaView is an open-source, multi-platform data analysis and visualization application available from Kitware Inc – [www.paraview.com](http://www.paraview.com).

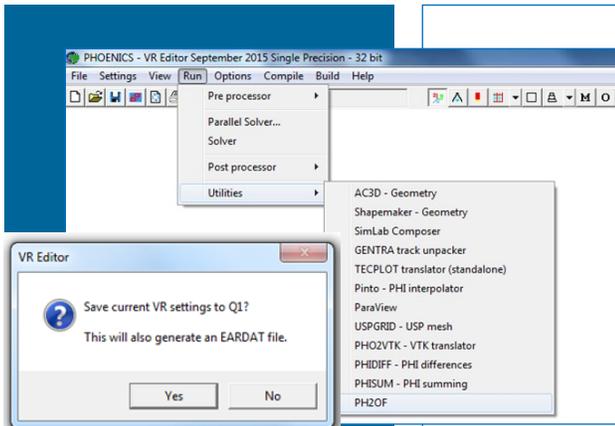
PHOENICS and OpenFOAM deal with faceted objects using different approaches. PHOENICS uses the PARSOL cut-cell technique, which automatically takes account of faceted objects on a structured grid.

OpenFOAM adapts the VR-generated grid to fit the contours of a faceted object. There is an option to refine the grid near faceted object surfaces by setting the appropriate refinement level. This is achieved by setting the variable, ISG1 in the ‘PIL Command’ box of the ‘GROUND’ tab within the PHOENICS VR-Editor (e.g. ISG1=2 for 2 levels of refinement). See below.

On completion of the case set-up, clicking on the menu item ‘Run/Utilities/PH2OF’ will execute the runOpenFOAM\_Residual.bat script. The VR-Editor will ask for confirmation to save the EARDAT file. Click ‘yes’ to ensure that the EARDAT file exists or is up-to-date.

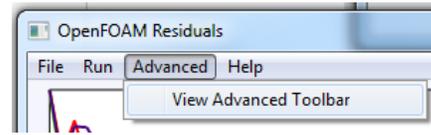
The batch script will run the PH2OF translator, producing the required OpenFOAM case files, and run the OpenFOAM solver whilst simultaneously plotting the residuals in the ‘OpenFOAM Residuals’ pop-up window.



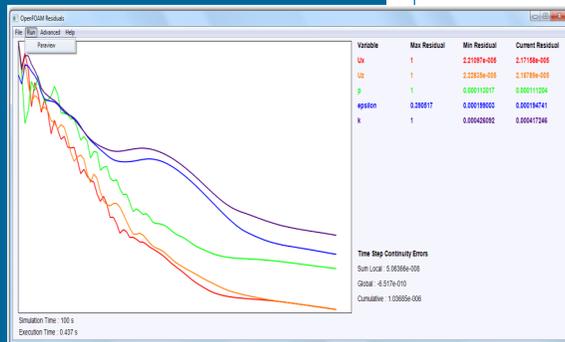


The OpenFOAM simulation can be aborted at any moment by right-clicking the mouse within the residual plot pop-up window. This will also write-out the final time-step data.

OpenFOAM files can be viewed via menu item 'Advanced/View Advanced Toolbar'.



This opens a side-toolbar with the option to view PHOENICS files. Double-clicking on a file item within the directory tree will enable it to be viewed within the window.

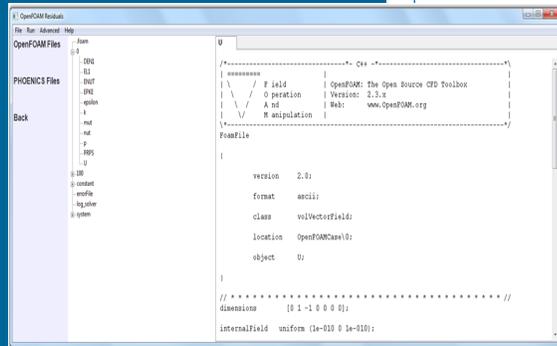


The 'errorFile' displays any errors occurring with the PH2OF translation process. The various log files produced by OpenFOAM, such as 'log\_solver', contain information relating to OpenFOAM simulation and are helpful in diagnosing any run-time errors.

## 2.2 Data visualisation

Results can be viewed in ParaView by clicking 'Run/ParaView' within the residual plot window. Alternatively, for simulations which contain non-faceted objects, a PHOENICS phi file will have been written to the working directory. This file can be viewed within the PHOENICS VR-Viewer by selecting 'User-set file names' and ensuring 'Solution file' is set to 'phi'.

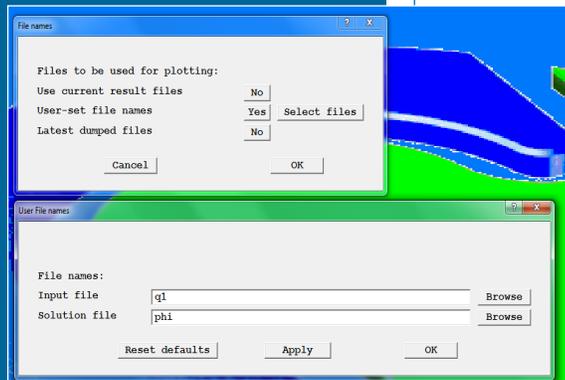
Note: At present, for simulations which contain faceted objects, results can only be viewed in ParaView. This can also be achieved by selecting the VR-Editor menu option 'Run/Utilities/ParaView,' and within the ParaView environment opening the file OpenFOAMCase/.foam to view current data.



## 2.3 Using the command line

PH2OF can be run without the need to enter the VR-Editor. In a working directory containing a valid EARDAT file, running the script 'runOpenFOAM\_Residual.bat' will cause PH2OF to generate the required OpenFOAM input files and then run the simulation whilst simultaneously plotting the residuals.

If an OpenFOAMCase directory already exists in the current directory it will be appended with the current date/time and another OpenFOAMCase directory will be created for the current simulation. Running PH2OF in this way, it is important to ensure that any faceted objects have been saved in STL format within the working directory and that the probe location has been positioned within the fluid region of the grid (by setting i,j,k values of the probe (IXMON, IYMON, IZMON)).



## 3. Files created by PH2OF

PH2OF will create a new directory called 'OpenFOAMCase' where all OpenFOAM case files are stored. In addition, batch scripts are created within the working directory to enable automatic running of simulations and post-processing of results. Any errors that occur during execution of the translator will be written to 'OpenFOAMCase/errorFile' as well as outputted to the console window during execution. Typical errors may be incompatible with thermophysical models or invalid turbulence model selection.

## 4. PH2OF Current Functionality

### Geometry

- Cartesian co-ordinate systems.
- Time dependence – Steady and transient.

### Objects

- Blockage
- Inlet
- Fan – Rectangular fan only
- Outlet
- Plate
- Porous plates – Superficial velocity only, with pressure drop formulation (Velocity-squared, Power of velocity, Linear in velocity).
- Thin-plate
- Pressure-relief

### Models

- One-phase only
- Scalar equation method for free-surface flows.
- Thermal modelling including conjugate heat transfer.
- Turbulence Models – Laminar, constant-effective, KEMODL, KOMODL.
- Transport of C1, C2 etc.
- No radiation at present.
- No combustion/chemical reaction at present.

### Properties

- The majority of material properties can be translated from PHOENICS into OpenFOAM. However, certain configurations are not compatible. An error is outputted for incompatible OpenFOAM material properties and available options are suggested. (e.g. In OpenFOAM, if the density is defined by the ideal-gas law then the viscosity must be defined by Sutherland's formula.)

### Sources

- Gravity – Constant, Boussinesq.uj

### Numerics

- Linear relaxation only
- Upwind differencing scheme. Others, such as, QUICK, VANL1, LUS, UMIST, CDS can easily be included.

### Helpful hints

- OpenFOAM often requires relaxation on pressure (P1) for convergence.
- Running PH2OF again in the same working directory will cause the old 'Open FOAM Case' folder to be appended with date and time (eg 'OpenFOAMCase\_08Jan16-142755'), and a current 'OpenFOAMCase' directory to be created.
- To diagnose errors first check 'OpenFOAMCase/errorFile'. If no errors are reported, check OpenFOAM log files, such as 'log\_solver' or 'log\_snappyHexMesh'.

## 5. Setting up OpenFOAM and PH2OF

- 1) Unzip 'OpenFOAM.zip' to the C:\ drive
- 2) Run MSMPISetup.exe
- 3) Install ParaView using ParaView-4.1.0-Windows-64bit.exe
- 4) Set the OpenFOAM and PH2OF environment variables by double-clicking on OpenFOAM-2.3.x-reg.
- 5) Add PH2OF menu option to VR-Editor by editing C:\phoenics\d\_allpro\ phoesav.cfg . Add the following line below the last entry of '\* List of Executable Files' UTILITY PROG = C:\OpenFOAM\PH2OF\ runOpenFOAM\_Residual.bat 'PH2OF',

## 6. Acquiring the PHOENICS-to-OpenFOAM translator

PH2OF will become available to existing licensees of PHOENICS-2015 and as a download via CHAM's FTP site.

Contributions to the Newsletter from all PHOENICS Users are welcome.

Please send articles, in Word format, to news@cham.co.uk.

Academic Users please send your annual reports to the above email address for publication.

Thank You.



## What's New in PHOENICS: Features added since October 26 2015 Release

John Ludwig , CHAM

### Solver – Earth

#### Solution of Scalars in Solids

Scalar solution and display (except TEM1, TEM2, H1, H2, EPOT, STORED scalars representing physical properties) are normally suppressed inside solids. It has also been possible to display derived variables inside solids if the variable NAME started with '#', eg #C1. It is now possible to solve for such variables.

VR-Editor has been upgraded to allow sources for such scalar variables to be set at participating blockages and at plates.

#### Reporting of KE and EP Sources

The source of KE and EP in the two-equation turbulence model is applied through the KESOURCE patch. Default linearization, KELIN=3, splits the source into linearised and non-linearised parts. The linearised part only was reported in 'Sources and Sinks' in RESULT, making it hard to judge the degree of source balance. The non-linearised part is now reported.

#### Auto-creation of CSV files from PROFIL patches

A PATCH command with type PROFIL can be used to create a table of values and a line-printer plot in RESULT. The table of values is now automatically written to a CSV file at the end of the run, making plotting profiles in Autoplot or Excel easier. For time history profiles, the X- axis is now the actual time at the end of each time step not the step number.

In parallel, data are written by the processor with the patch. Distance plots straddling processor boundaries are written on each crossed processor; time histories appear on the processor the patch is located in.

For distance plots in transient cases a separate file is written each print step with the step number written after patch name eg prof000101.csv.

If the patch is linked to a VR object, e.g. a POINT\_HISTORY object, the object name is used as the base of the file name since the patch name will be meaningless to users.

#### Corrections to Scalar Equation Model (SEM)

In the Scalar Equation Model, convection terms of the SURN equation do not contain density. Erroneously, the convection terms of other SOLVED variables also did not include density. This is fixed.

Reporting residuals of SURN in the Scalar Equation Model is improved.

#### General Improvements and Corrections

If the RESULT file is held open in another program when the Earth solver tries to write to it, a warning is issued and users are given the option to exit the other program and retry. Previously the Earth run would crash.

Improved 1-D block corrections in parallel to prevent occasional NaNs.

If a transient run is ended on a step which is not a printing step, the residual table written to RESULT contained 1E19 instead of the residuals for that step. This is corrected.

If a run fails it is difficult to analyse why without a phi file. A phi file, starting with a # symbol, will be written whenever failure is detected. The ability to set diffusivity for velocity equations using InForm, for example (Property of PRNDTL(U1) is -1) has been added.

Some cases using PARSOL can take a long time to start calculating. The algorithm causing this has been corrected.

## News From Champion: Taiwan: sales@cpet.com.tw



Champion held a PHOENICS Seminar for CTCI Co in Taiwan in December 2015 on the application of PHOENICS in civil engineering & eco-environments. Photos above.

## News from Shanghai Feiyi: China: sales@shanghaifeiyi.cn

- 1) Shanghai Feiyi exhibited at the Twelfth international Conference on Green & Energy-Efficient Building & New Technologies and Products in Beijing March 30-31 2016.
- 2) A PHOENICS-2015 Conference and User Meeting will be held end March / beginning April 2016.
- 3) A PHOENICS User Meeting will be held in Guangzhou where an office will be set up during April 2016.
- 4) A PHOENICS User Meeting will be held in Beijing during April.



## VR-Editor

### General Improvements and Corrections

Small dimension cases can now be viewed without the need to click 'Fit to Window' to avoid their appearing as a blank screen. When reading the near-plane position from Q1.

A formula error used to set the initial EP profile for the WIND object when the inlet profiles are read from a table has been fixed by adding missing (). The initial EP now reads  $CD/AK*(PWLF(file,ZG)^{1.5})/ZG$  not  $CD/AK* PWLF(file,ZG^{1.5})/ZG$ .

The rotation centre is now initialised correctly so there is no unusual behaviour when newly-created objects were added to an assembly object.

ASSEMBLY objects are created in wireframe mode so contents can be seen.

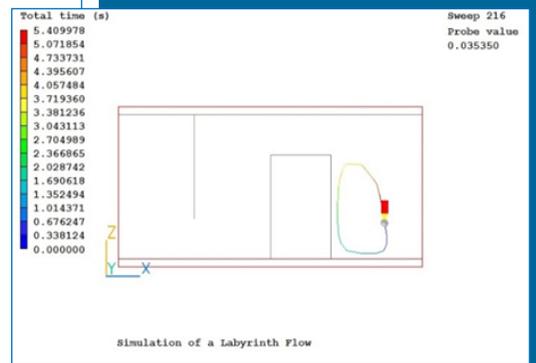
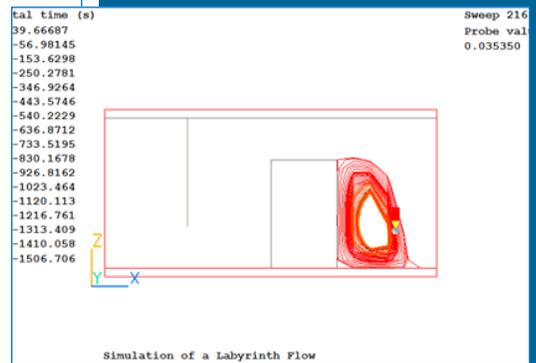
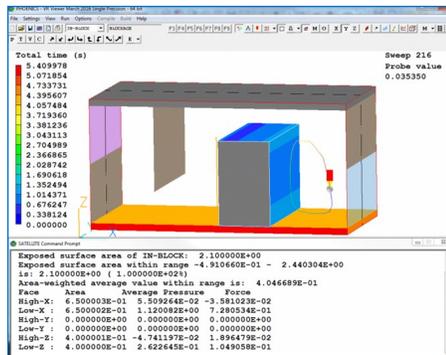
## VR-Viewer

### Plotting Closed-loop Streamlines

If a streamline passes through the cell containing the start-point of the track, and the track then passes with half a cell size of the start-point, the track is terminated and drawn as a closed loop. The first image on the right shows the previous streamline plot, the second the new closed form.

### Average Values

The area-weighted average value of a variable over a surface can be obtained by turning on 'Surface contours' for any object, and then 'Dump surface values'. The image below shows a typical output:



### General Improvements and Corrections

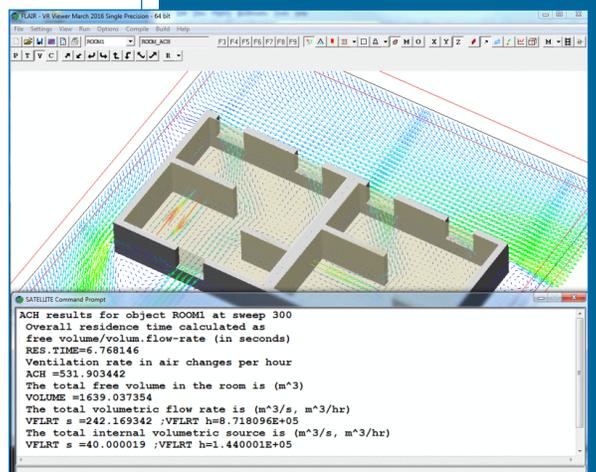
Corrections to do with how often surface contours are regenerated, to prevent unnecessary contour regeneration when moving the probe.

The VRV macro command to plot a line plot caused the execution of an infinite loop, making VRV freeze. Further, the macro command to dump an image file of a line plot did nothing. These are fixed.

## Flair / Flair-EFS

### Air Changes per Hour for a closed volume

A ROOM\_ACH object has been introduced which defines a closed volume eg a room within a building. The sum of volumetric inflow rates through the object faces is computed and divided by the free volume within the object. This is multiplied by 3600 to give Air Changes per Hour for the object. The ACH value is printed to RESULT, and can be displayed in the Viewer by right-clicking a ROOM\_ACH object and selecting 'Show net sources'. The image on the right shows a typical usage for flow through a building:



Two ROOM\_ACH objects are seen, one per room. As the objects align with room walls, flow rates calculated are inflows through windows. The flow from internal inlets is also captured.

### Apparent Temperature – Heat Index

Two forms of Apparent Temperature have been added based on a mathematical model of an adult *walking outdoors* in the shade. The version including temperature, humidity, and wind effects is:

$$AT = Ta + 0.33 \times e - 0.70 \times ws - 4.00$$

The version including temperature, humidity, wind, and solar radiation effects is:

$$AT = Ta + 0.348 \times e - 0.70 \times ws + 0.70 \times Q / (ws + 10) - 4.25$$

where:

Ta = Dry bulb temperature (°C) (PHOENICS TEM1)

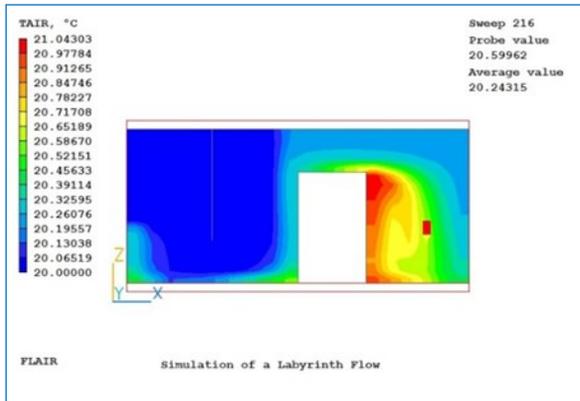
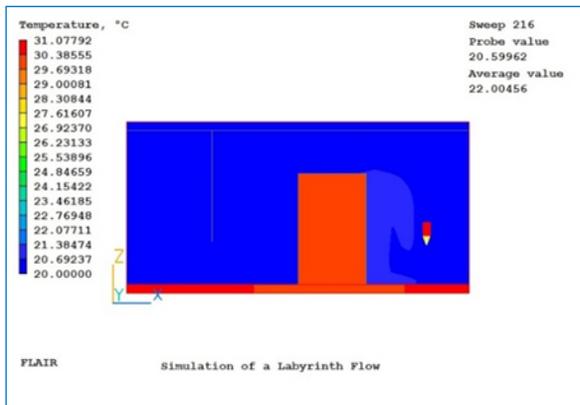
e = Water vapour pressure (hPa) [humidity] (PHOENICS PVAP)

ws = Wind speed (m/s) at 10 metres elevation (PHOENICS VABS)

Q = Net radiation absorbed per body surface unit area (w/m<sup>2</sup>)

### Air Temperature (not including Solids)

The variable TAIR has been added to the Comfort Index panel. This is just the temperature, TEM1, but only in cells containing fluid. The images below show the normal temperature, TEM1 at the top and the air temperature, TAIR, at the bottom:



## Brief News from Abroad

### News from CHAM Japan: 2016

Seminar Dates: [customer@phoenics.co.jp](mailto:customer@phoenics.co.jp)

Regular Seminars are held in Japan. Please contact the email address above to find out future Dates and Topics.

### News from A2TE :

Benelux Countries: [gjanssen@a2te.nl](mailto:gjanssen@a2te.nl)

A Benelux User meeting will be held in the Summer of 2016 in Eindhoven. More information will be provided in future Newsletters.

### News from ArcoFluid :

France: [arcofluid@arcofluid.fr](mailto:arcofluid@arcofluid.fr)

PHOENICS in the field of Micro fluidics was presented at a meeting of CGS Lab January 27 & 28 2016 at the JF Centre, TOTAL , Pau, South West France.

PHOENICS in complex flows will be the subject of several lectures and seminars at the University of Marseille, IRPHE Château-Gombert from mid-April to mid-May 2016.

### News from Safe Solutions: Brazil:

[fabio.fundo@safesolutions.com.br](mailto:fabio.fundo@safesolutions.com.br)

On January 29 2016 Safe Solutions held a workshop at Jaragua do Sui to present PHOENICS to some refrigeration companies of the region and also to three local Universities.

### News from SSC:

Russia: [alex@mons.mei.msk.ru](mailto:alex@mons.mei.msk.ru)

Two PHOENICS- related lecture courses will be held at MPEI in Moscow. More information will be provided in future Newsletters.

Contents	Page
Simulation of a New CSO Chamber Design for the Water Industry	2
Modelling Crowd Behaviour Using PHOENICS	4
PHOENICS to OpenFoam Interface	7
What's New in PHOENICS 2015	10
News Items	10, 12

