

# PHOENICS

## News

Summer / Autumn 2009



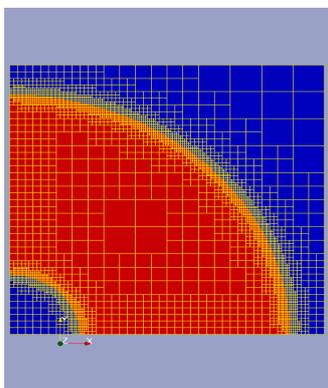
### 1) Editorial

Professor Brian Spalding, FRS, FEng and Foreign Member of the Russian Academy of Sciences, received his Global Energy International Award from President Dmitry Medvedev on June 5 2009 during the International Economic Forum held in St Petersburg ([www.forumspb.com](http://www.forumspb.com)).



Professor Spalding, who was honoured for “numerous original conceptions in the processes of heat and mass transfer which have become the basis for practical calculations of the mechanics of fluid flow”, gave an acceptance speech during the award ceremony. The Forum was attended by economists worldwide and both it, and the award ceremony, received extensive coverage in the Russian Press (link to video). (<http://www.ge-prize.ru>).

One of Professor Spalding’s duties as a Laureate is to deliver five lectures on topics associated with Global Energy during the year from June 2009 to June 2010. His first lecture, to start fulfilling this obligation, was on “Simultaneous Solid-Stress and Fluid-Flow Computation” given at St Petersburg Polytechnic University on June 3 ([www.cham.co.uk/laurlec.ppt](http://www.cham.co.uk/laurlec.ppt)).



Automatically-generated unstructured grid for computing stresses within a hollow cylinder (red) created by differing pressures in the inner and outer fluid regions (blue).



Professor Spalding has also lectured on “Unstructured PHOENICS for Wind-Farm Simulation” to a meeting at WindSim in Tonsberg Norway (June 17) ([www.cham.co.uk/usp.ppt](http://www.cham.co.uk/usp.ppt)) and on “Advanced Stream Analysis: Predicting the Flow in Shell-and-Tube Heat Exchangers” to the UK Heat Transfer Society in London on June 24 ([www.cham.co.uk/shellflo.ppt](http://www.cham.co.uk/shellflo.ppt)). In the latter lecture Professor Spalding presented his work on SHELLFLO (described in the last Newsletter) which provides a single-purpose Gateway to PHOENICS for use by designers of, and those interested in, Shell-and-Tube Heat Exchangers.

The next lecture, “Population Models of Turbulent Heat and Mass Transfer” ([www.cham.co.uk/rome.ppt](http://www.cham.co.uk/rome.ppt)), will be given at the 6<sup>th</sup> International Symposium on Turbulence, Heat and Mass Transfer to be held in Rome from September 14 – 18 2009. The Symposium is organised by ICHMT (International Centre for Heat and Mass Transfer) which was founded by Professor Spalding and a group of like-minded scientists and sited, originally, in the then Yugoslavia, to provide a base where scientists from both East and West could meet and exchange scientific views. (<http://www.thmt-09.org>).

TABLE OF CONTENTS		Page
<b>Section 1</b>	<b>Editorial</b>	1
<b>Section 2</b>	<b>Recent PHOENICS Features</b>	
2.1	Unstructured PHOENICS	2
2.2	Automatic Unstructured Grid Generator	2
2.3	Other Features	3
<b>Section 3</b>	<b>PHOENICS Case Studies</b>	
3.1	Thermal Management of Outdoor Enclosures using PHOENICS	6
3.2	Numerical Analysis of a Flame in a Flare exposed to Crosswind	8
3.3	Numerical Analysis of Atmospheric Air Flow Entrainment in a House Stove Mixer	8
<b>Section 4</b>	<b>Consultancy Applications</b>	
4.1	Data Centre Simulation for Thomson Reuters UK	9
<b>Section 5</b>	<b>Agent Applications</b>	
5.1	Chemtech	10
5.2	ACFDA, Canada & USA	11
<b>Section 6</b>	<b>News &amp; Events</b>	11

**2. Recent PHOENICS Features**  
**2.1 Unstructured PHOENICS by Brian Spalding**

Although Unstructured PHOENICS has been operational since mid-2007, it has not so far been described in a Newsletter article. Now however, particularly because its use has been facilitated by the introduction of an Automatic Grid Generator, it is time so to do.

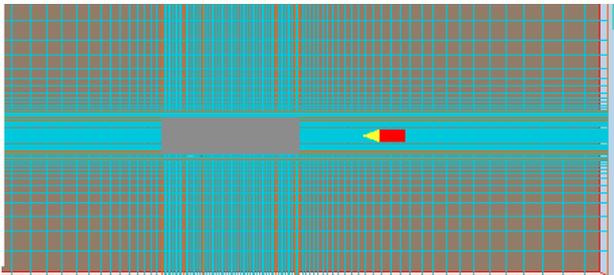
**General features**

Unstructured PHOENICS (USP) is **not** a separate software package; instead it is a part of the Solver module, EARTH, to which it is attached as an option. USP is activated, like any other PHOENICS option, by placing appropriate instruction lines in the Q1 file, which is read by the standard SATELLITE module. Therefore its user has very little new knowledge to acquire.

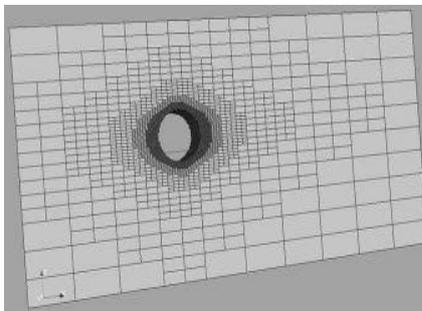
That is not quite true of the display side; for neither PHOTON nor the VR-Viewer has yet been taught how to work with an unstructured grid. Therefore graphical display has to be effected by way of packages such as the public-domain code Paraview or the commercially available Tecplot; for USP expresses its results in files of Vtk format (as can now, indeed, structured PHOENICS).

**The reasons for introducing USP**

The **motive** for introducing an unstructured grid has **not** been, as for some other CFD codes, to enable PHOENICS to handle **curved-surface** bodies; for its 'cut-cell' technique, **PARSOL**, copes with these satisfactorily. Instead, its purpose is to promote **economy** of computer time and memory. Many users of the standard structured code will have sometimes had to use grids like the one below. Needing a fine grid around a body located in the central region, they have been forced to create cells of needless fineness, and of poor aspect ratio, in regions remote from the body.



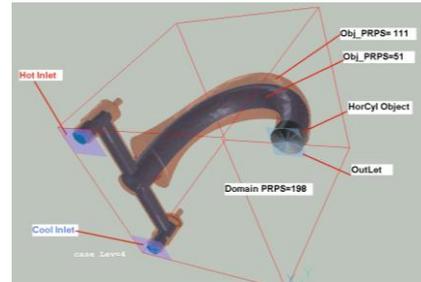
USP removes this necessity, as is shown below: the fine-grid region hugs the surface of the central body, which is where it is needed for procuring accuracy; farther from the body the cells are larger, and also of acceptable aspect ratio.



As can now be seen, USP cells are always rectangular, and of uniform aspect ratio. Although vendors of other CFD packages often employ unstructured grids in which the cells are of arbitrary rectangular shape, CHAM sees no significant advantage in so doing.

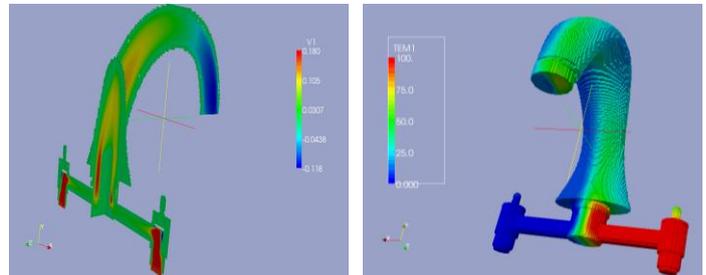
**An example: the hot-and-cold mixing faucet**

There are many examples of fluid- and heat-flow simulation in which the region of interest occupies only a small fraction of the volume of the smallest rectangular box which could be drawn around its extremities.



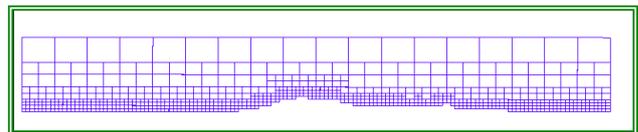
One such is shown above. Of course, it would be **possible** to simulate the fluid- and heat-flow phenomena by using structured PHOENICS. However, to do so with acceptable accuracy would be prohibitively expensive. The reason is that for every fine-grid cell in the water or the metal there would be hundreds or thousands in the air outside, performing calculations of no interest whatsoever.

USP, by contrast, pays no attention to the outside region; so it wastes neither computer time nor memory. Some results of its computations for this example are shown below.



**Application of USP to terrain-type problems**

USP is particularly useful for flow-over-terrain problems, where fine grids are required near the ground, whereas coarser ones suffice for higher altitudes as is shown below. Wind-farm-simulation studies can benefit from its use.



**2.2 The Automatic Unstructured-Grid Generator (AGG) by Brian Spalding**

**The Main Features**

Unstructured PHOENICS (USP) has been made easier to use by the introduction of an Automatic Grid Generator module, referred to as AGG below.

AGG starts by constructing a coarse uniform Cartesian grid, which fills the whole domain; then, in accordance with simple instructions which the user places in the Q1 file, it systematically refines the grid in the vicinities of solid objects, inlets, outlets and other features of the flow-simulation scenario.

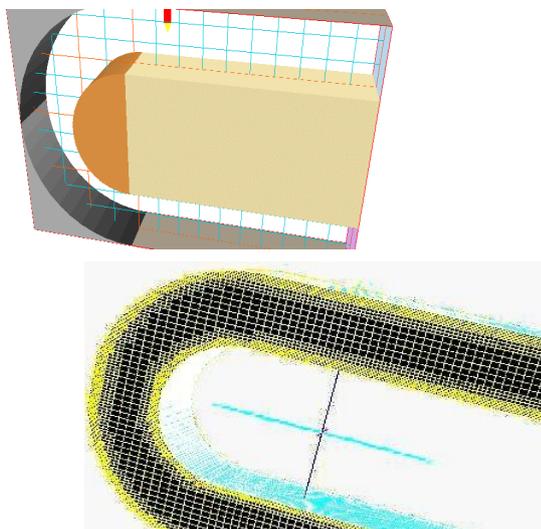
AGG can be used for two-dimensional and three-dimensional scenarios. In the former case, refinement proceeds by sub-dividing individual cells into **four** equally-sized smaller ones; in the latter case the number of smaller cells is **eight**.

Instructions given by the user dictate how many levels of sub-division are provided and how many cells of a given size should be placed in each layer of equally-sized cells.

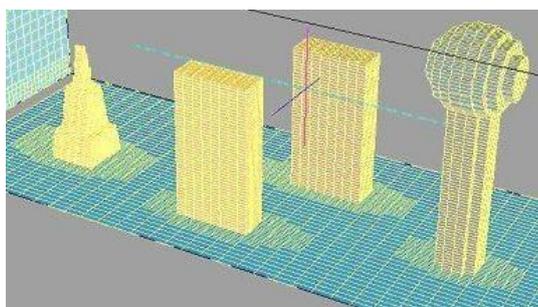
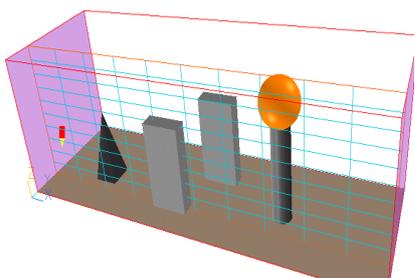
### A Two-Dimensional Example

The following two diagrams concern the 'turn-around duct' which is used in the PHOENICS library to illustrate the benefits of the cut-cell technique, PARSOL.

The first one shows the coarse initial grid; on the right is shown the grid which AGG created, with 4 levels of refinement and 4 cells per layer.



Next is shown the near-surface grid refinement of a three-dimensional example, with several differently-shaped objects.

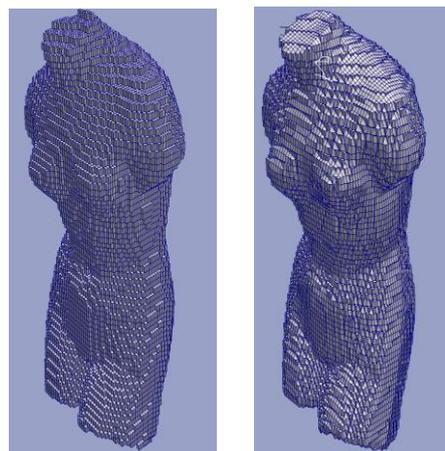
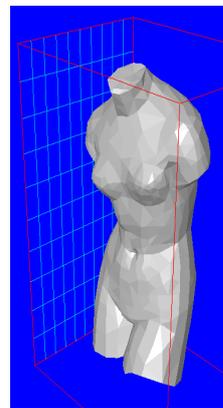


*The Near-Surface Grid*

### The Smoothing Algorithm

Because AGG can place many small cells close to surfaces, unstructured PHOENICS is less in need than is structured PHOENICS of the cut-cell technique to improve simulation accuracy. Nevertheless AGG provides something of the kind, namely a Smoothing Algorithm which changes the shapes of the cells near solid bodies, so as to fit as closely as possible. Its purpose is more to improve visual display than to enhance accuracy.

Because it is constrained by the requirement that each three-dimensional cell must still have six faces, the smoothing is not perfect; but it always effects some improvement, as shown by the following three pictures, of which the first shows the shape represented by facets, the second the near-surface grid without smoothing and the third the surface grid **with** smoothing.



### Further developments

Although AGG was designed with the fitting of solid bodies in mind, it has proved to be more versatile. A recent development is its extension to solution adaptive re-gridding,

For example, after the solution has proceeded to near-convergence with an initial grid, AGG can be invoked again with instructions to provide a finer grid in regions of (say) highest temperature gradient. This capability offers the possibility of improved accuracy of simulation of flame fronts, for example.

## 2.3 Other Recent PHOENICS Features

*by John Ludwig*

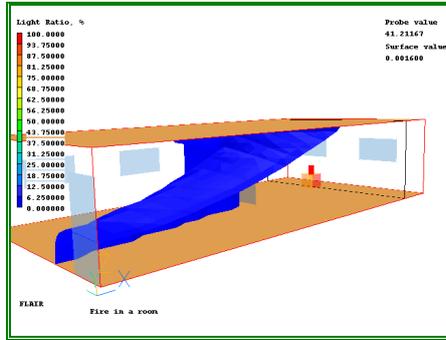
### 2.3.1 Flair Updates

A list of fire speeds for the time-squared fire has been added so that slow, medium, fast and ultra-fast fires (as defined in Table 10.1 of the CIBSE Guide E, Fire Engineering) can be simulated without having to look up the values of the relevant constants. The user can also input their own constant if they want.

A list of fire speeds for the  $t^2$  fire has been added so that all fires from Table 10.1 of the CIBSE (Chartered Institute of Building Services Engineers) Guide E, Fire Engineering can be simulated.

An explicit allowance for [heat lost by radiation](#) is made when setting the FIRE heat source. This allows the full heat-release-rate to be used in deducing the mass source, whilst allowing the convective heat release to be reduced (by 1/3) in line with guidelines.

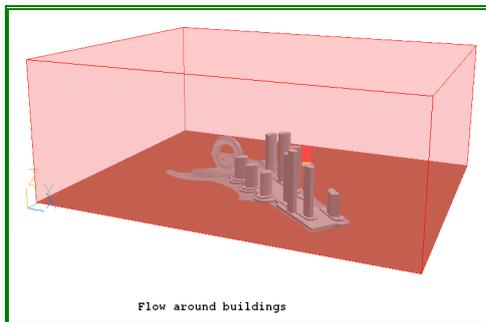
The [Beer-Lambert visibility reduction](#) can be calculated as a post-processing option in the Viewer. This is a means of determining if a position can be seen from other locations, or if it is obscured by smoke. The image below shows that a fire would be visible from the foot of the door at the left, but not from head-height.



The linearisation of the fire heat source has been improved to aid convergence of the temperature equation with large heat sources. The maximum increment for TEM1 is automatically reduced when a FIRE object is active which should make such calculations more stable.

### 2.3.2 Pre- and Post-Processing

#### 2.3.2.1 WIND Object (see also Spring Newsletter)



The wind object, introduced in late versions of PHOENICS 2008, has been further enhanced:

1. It can now create the floor plane as the flow and pressure boundaries, using the same roughness height as was set for the velocity profile. This saves the user from having to create the ground as a separate object, and ensures that the roughness settings for the ground are correct.
2. If wind direction is at right angles to a domain face, the two side faces are also treated as pressure boundaries, as is the opposing face. Previously the side faces were treated as symmetry planes, which may have constrained the flow.
3. If wind speed is set to zero, all faces of the domain are set as pressure boundaries providing an easy way of placing pressure boundaries on all open faces of the domain.

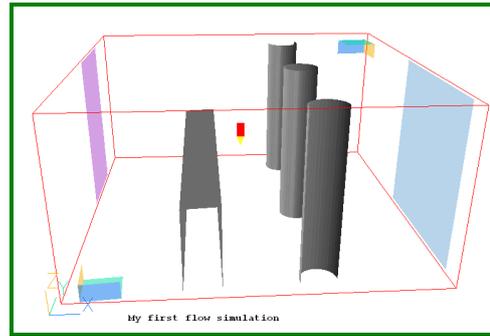
#### 2.3.2.2 Ambient Pressure and Temperature

The 'Ambient pressure' and 'Ambient temperature' entries set the pressure and temperature prevailing outside the domain. These values can be used, if desired, to set the external pressure and temperature at all [INLET](#), [WIND](#), [WIND PROFILE](#), [OUTLET](#), [FAN](#) and [PRESSURE RELIEF](#) objects.

When 'Initialise from ambient' is set ON (default), the initial values of pressure (P1) and temperature (TEM1, and T3 if IMMERSOL is on) are always made consistent with the ambient values set here. When 'Set buoyancy from ambient' is ON (default), the reference temperature for [Boussinesq](#) buoyancy is set to the ambient temperature, or for [density difference](#) buoyancy, the reference density is calculated from the ambient pressure and temperature.

This makes it much easier to change external temperature/ pressure at many boundaries in a consistent fashion. It also ensures that the buoyancy reference density is more likely to be set correctly.

#### 2.3.2.3 Clipping Plane Object (see also Spring Newsletter)

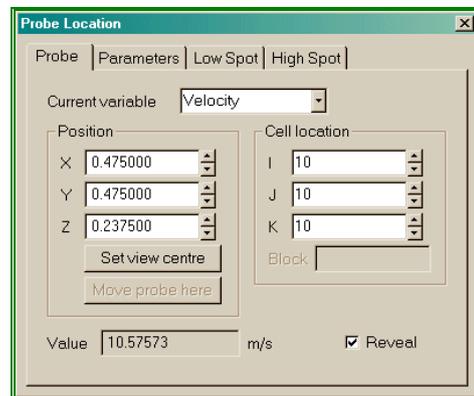


Each **Clipping\_plane** object comprises 3 OpenGL clipping planes. A maximum of two objects are allowed, as there are only 6 clipping planes. The first clipping-plane object (the left-hand object above) is created at origin and clips everything 'behind' it. The second clipping plane object is of type '**High-end clipping plane**' (on the Options tab) and is originally located at the far end of the domain (the right-hand object above). It clips everything 'after' it. These can be moved and rotated at will to provide a better view inside objects.

Clipping plane objects can be created from '[Settings - New - Clipping plane](#)' on the [Environment Settings](#) menu, or '[Object - New - Clipping plane](#)' on the [Object Management Dialog](#) in the Editor or the Viewer. When created in the Editor, they will be written to the Q1 like all other objects. When created in the Viewer, they will disappear when the Viewer is exited. In the Viewer, the user can choose whether they clip the plotting plane or not. Clipping plane objects never affect the grid, and have no influence on the solution.

#### 2.3.2.4 Set View centre to Probe / Minimum / Maximum value

Double clicking on the probe or clicking the  icon on the toolbar causes the following dialog to appear:



It contains a summary of data at the current probe location. In Viewer, the current variable can be changed via the pull down list. Probe location can be moved through the model in physical space (X,Y,Z position), or by cell location. When moving in physical space, 'Cell location' boxes will show the cell centre nearest the probe. When moving by Cell location, the X,Y,Z boxes will show the physical location of the probe. The '**Set view centre**' button centres the view on the probe. In the Viewer, Low Spot and High Spot tabs have similar buttons, and the 'Move probe here' button is active. This allows highest and lowest values to be located and viewed quickly

#### 2.3.2.5 Patches for User Defined Objects

Patches attached to USER-DEFINED object are listed by name as part of the object attributes in the Q1 file. The PATCHES attribute contains a list of the patch names associated with an object. As many PATCHES lines as are needed to hold all patch names can be used.

- > OBJ, TYPE, USER\_DEFINED
- > OBJ, PATCHES, patch1, patch2, patch3, patch4, patch5
- > OBJ, PATCHES, patch6, ... patchn

All settings relating to PATCH and COVAL statements linked to a user-defined object are printed in the relevant Group - these can be Groups 11, 12, 13, or 23. The name of the controlling object is written as a guiding comment.

The first PATCH location argument (usually IXF) is set to -1, to indicate that the patch is to be linked to an object. The remaining five location arguments are zero. As many PATCH commands as required can be attached to one user-defined object.

In earlier (pre-2009) versions the IXF location argument was used to hold the object number. This method is still recognized on reading a Q1, but when a new Q1 is written the object number will be replaced by -1, and the patch name will be echoed in the PATCHES list. This makes hand-editing the Q1 much safer, as the patch - object link no longer depends on object sequencing which is easily lost when moving objects by cutting and pasting.

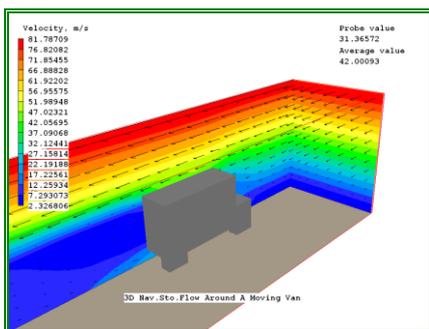
**2.3.2.6 Inform Commands attached to Objects** (see also *Spring Newsletter*)

Many objects types now have a button labelled 'InForm Commands' on their attributes page. This leads to a dialog from which a selection of [InForm](#) commands can be attached to the current object.

The dialog allows such commands to be created with the LOCATION keyword being taken as the name of the current object. InForm commands created in this way are [held in the Q1](#) together with the remaining object attributes. As an example, the image below shows the settings required to make the inflow mass source and velocity at an INLET a linear function of the height.



The resulting flow is:



If an object is copied/arrayed Inform commands are copied.

**2.3.2.7 Sweeps Per Time Step**

The sweep number per time step can be set from Editor Main Menu Numerics panel (always possible with InForm). In many transient cases more sweeps are needed for the first few steps and possibly later if a boundary condition changes - a jetfan switches on or off for example.

**2.3.2.8 General Improvements: Information on these features can be obtained via links show:**

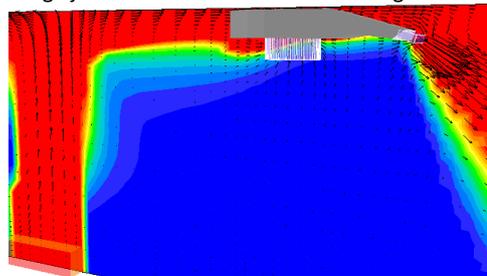
- [SPINTO](#) to interpolate solution files on finer/coarser grids.

- [Automatic unstructured-grid generator](#) for unstructured PHOENICS refines automatically an initial structured starting grid to improve resolution near object surfaces (see page 4).
- [SHELLFLO](#), a Prelude gateway for shell-and-tube heat exchangers.
- [Dot-patches](#) introduced. This is a way of setting the patch limits in terms of physical location and size, not cell numbers.
- View centre can be jumped to current probe position, or location of minimum or maximum value in Viewer, making it much easier to find the minimum and maximum value locations.
- Creation of complex InForm commands has been made easier by increasing the length of a Q1 line from 132 characters (68 characters and one continuation line) to 1024 characters split over as many continuation lines as needed.
- Viewer display can be switched to FPS, cgs or SI units (default).
- Domain scaling is useful for long thin domains, ie tunnels but not scaling vector heads improves plot appearance; therefore, when the domain is scaled, vector heads retain their original scaling.

**2.3.2.9 Linked ANGLED-IN Objects**

It is possible to take flow extracted by one ANGLED-IN and use it as input to another. Average temperature, smoke and other scalars at the 'donor' object are used as the inlet values for the 'target'.

This makes it possible to model Induction fans, or duct-work as pairs of linked ANGLED-Ins. The next image shows the smoke ingested through the ANGLED-IN on the lower face of the induction fan body being ejected from the ANGLED-IN acting as the nozzle.



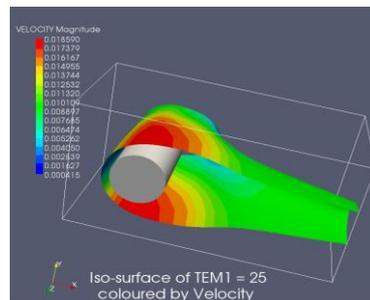
**2.4 Earth Solver**

**2.4.1 Output to Paraview**

The Earth solver can now output files in [VTK](#) format. These files are compatible with the [ParaView](#) post-processor available for free download on the web. In total, two files are involved.

- vtkcentre.vtk
- vtkvertex.vtk

Vtkcentre.vtk contains a grid located at the PHOENICS cell centres. The data is the 'raw' PHOENICS data, apart from the velocities which have been averaged from the cell faces to the cell centres. For PARSOL cut cells, the cell centre locations are the centres of the fluid cells. This zone is suitable for plotting vectors, as the vector tails will be in the same location as in the Viewer.



Vtkvertex.vtk contains a grid located at the PHOENICS cell corners. The data has been averaged from the surrounding cell centres for scalars, or cell faces for vector quantities. The values in PARSOL cut cells are those from the fluid cells. Contours plotted from this zone will fill to the edge of the domain as in the Viewer.

To display the objects, Editor can output STL files of each one (Options, Additional Interfaces, Additional Individual STL output) which can be loaded into ParaView. The above image shows an iso-surface of temperature coloured by velocity from Library case v146, produced by ParaView from the vtkvertex.vtk file. The cylinder is drawn using the STL file produced by Editor.

**2.4.2 Nett source printout**

The 'Nett source' section of RESULT includes average scalar values at mass source patches. For pressure boundaries, inflow and outflow are given as well as the nett flow. For transient cases, the transient contribution to the mass sources is printed, so the mass sources should balance.

**2.4.3 Changing the number of sweeps during a run**

The sweep number can be changed at Earth runtime from the 'Reset' screen of the graphical convergence monitor which is particularly useful in transient cases. The sweep number can be increased if the step is not converging well, or decreased if it has converged. If a steady run is not converged more sweeps can be performed without having to stop.

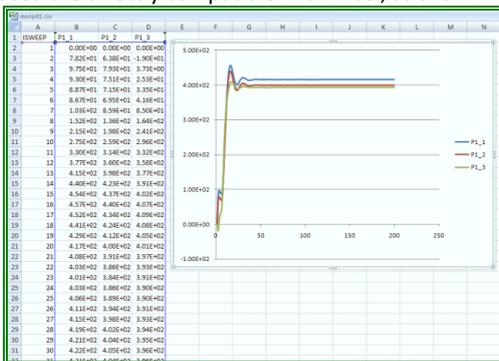
**2.4.4 Input-output matters**

PBCL.DAT file format has been changed so the writing process is more efficient. When PHIDA=T in PREFIX, PBCL.DAT is also unformatted. Old-style PBCL.DAT files are still accepted by Earth and Viewer. PHIDA=T is now the default making saved files smaller and quicker to read and write.

If the main F-array size is increased at the start of a run, the increased size is written to a local CHAM.INI file to remove the need for expansion on the next run.

**2.4.5 Tabular Output from InForm**

A new TABLE command has been added to InForm, which allows users to create sweep-wise or timestep-wise tables of any variables. This can be used to create additional monitor points, track average or minimum/maximum values etc. Files produced are directly compatible with Excel, as shown here:



The image shows the variation of pressure with sweep at three points in Library Case 274.

For more details about any of the above click [here](#)

**3. PHOENICS Case Studies**

**3.1 Thermal Management of Outdoor Enclosures using PHOENICS by Maurice J Marongui, MJM Engineering**

**1. Introduction**

In the last few years there has been a remarkable development in outdoor applications as a variety of end-user services is increasingly being created. The housing of equipment outdoors has imposed serious constraints on enclosure design since temperature and humidity are the two

major causes of electronics failure. Since most systems do not include environmentally hardened designs, the enclosure must provide an environment in which they can survive.

To meet the demands of protecting all types of temperature/moisture sensitive equipment in the outdoor environment, designers have been pushing for sealed designs. These allow for little or no exchange of enclosure air with the outside air using near airtight construction, high-pressure door closure designs and closed loop thermal systems. The thermal management of the outdoor enclosure thus becomes an issue of paramount importance. It is not only necessary to remove the internal heat generation from the equipment but also the effects of the solar loading which can be substantial depending upon the size of the enclosure.

This article will show and explain the thermal issues (including typical solutions) using, in part, PHOENICS.

**2. Outdoor Enclosure Thermal Management System Design**

Outdoor enclosures are being designed to house various equipment configurations with dissipating heat rates ranging from 500 up to 10000 W, depending on the size and type of equipment, (see figure 1). These enclosures are being installed in various environmental conditions, and typically the enclosures, without major structural modifications, should be fitted with either air conditioning or air-to-air heat exchangers as needed.

The main goal of the designer is to maintain the peak temperatures in the enclosures below a certain level that is normally prescribed by the equipment manufacturer. Humidity levels are of concern, but since most enclosures are either sealed or its temperatures are much higher than the air's dew points, humidity is generally not a problem (after the transient effect of opening/closing the enclosure is eliminated.)

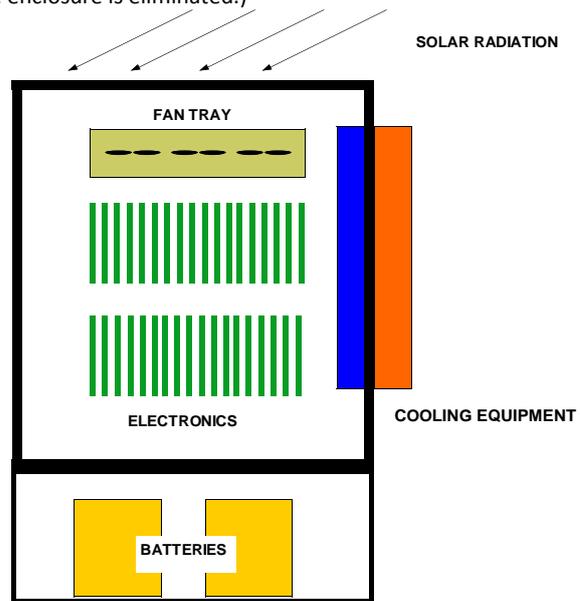


Figure 1 Typical Outdoor Enclosure

The designer should be aware that the air temperatures within the enclosures are a function of [1]:

- 1) Amount of heat generated by all equipment in the enclosure.
- 2) Amount of heat generated by auxiliary and cooling equipment (fans, etc.)
- 3) Ambient conditions (outdoor air), particularly temperature, solar radiation, wind speeds, etc.
- 4) Objects surrounding the enclosure (shading, ground reflections, buildings, trees, etc.)
- 5) Enclosure design (surface area, shape, paint's radiation characteristics, etc.)
- 6) Air exchange with the outside air, either passive by infiltration, or active, by fans or blowers.

Outdoor temperatures in most typical environmental conditions range from -40° C to 55° C

### 3. Cooling/Heating Load Calculations for Enclosures

We are now ready to begin the design of the thermal management system for a typical enclosure. Let us say that the enclosure to be designed is typical for the business, measures 2 m high and has a foot print of 3 m x 0.75 m. The enclosure has installed equipment that dissipates, say 1500 W, and must be kept at either the air-conditioning conditions or the air-to-air heat exchangers. The first step is always to realize that the design temperature is that temperature the enclosure air will attain when there is heat balance, or in equation form:

$$Q_{balance} = 0 = Q_{equipment} + Q_{solar\_load} - Q_{cooling\_system}$$

where,  $Q_{equipment}$  is the internal heat dissipation,  $Q_{solar\_load}$  is the solar heat load and  $Q_{cooling\_system}$  is the amount of heat removed by cooling system. The solar load is a complicated term because it includes contributions from all modes of heat transfer. For example:

$$Q_{solar\_load} = Q_{radiated} + Q_{convected} + Q_{conducted}$$

Normally, the value of  $Q_{radiated}$  will always be positive (towards enclosure) but the other two can be either positive or negative, depending on the enclosure's temperature. Thus, if  $Q_{balance}$  is not zero, this means that the temperature inside the enclosure is either higher/lower than the set temperature and the enclosure is losing/gaining heat by convection and conduction.

Solar load can be calculated using the Sol-air method as recommended by ASHRAE [2], which takes into account some convection and reradiation effects. Furthermore, since incident solar radiation varies during the daylight hours, the designer must decide whether to conduct a steady state or transient analysis. Moreover since  $Q_{radiation}$  is a very complex term that includes, among other effects, solar declination, latitude, time of year, solar azimuth, atmospheric absorption, atmospheric clearness, reradiation from other walls, buildings, ground etc., incident wall surface properties, some simplifying measures must be taken into account [3]. The result is that one can effectively double or triple the amount of heat flux being added into the enclosure depending on the calculation method. Normally, when calculating cooling loads, one would include a) Space heat gain, b) Space cooling load, and c) Space heat extraction rate [2]. Space heat gain is the rate at which heat enters or is generated within the space at any given instant. This includes for the enclosure heat transferred into the conditioned space from the external walls and roof due to solar radiation, convection and temperature differential.

One would normally include instantaneous solar radiation effects and delayed effects. The delayed effects include the slow build-up of energy that the external walls accumulate as they absorb solar radiation. This happens because walls are normally thick and massive; making energy absorbed important. For thin-walled enclosures this is not included. Another component of heat gain is latent heat due to moisture infiltration. For most sealed outdoor enclosures, the equipment is kept in an airtight enclosure with negligible contribution.

One method, the Sol-air temperature, involves calculating heat loads using an external temperature that lumps radiation effects and sensible air temperature. No attempt at taking into account solar inclination and radiation intensity variations during a day-cycle is made. One must understand that the enclosure's solar load is calculated for the worst condition. Hour-by-hour calculations can be carried out using BIN data but, in most cases, this is not necessary (since outdoor enclosures are not large, therefore they have low thermal mass).

Sol-air Temperature method, involves calculating heat loads using an external temperature that lumps radiation effects and sensible air temperature. This is expressed as [2]:

$$T_e = T_{out} + \alpha I_t/h_o - \epsilon \Delta R/h_o$$

where,  $\alpha$  -absorptance of solar radiation surface,  $I_t$  - total solar radiation  $[W/m^2]$ ,  $h_o$  - coefficient of heat by long wave radiation and convection  $[W/K-m^2]$ ,  $\epsilon$  - hemispherical emittance, and  $\Delta R$  a radiation correction factor  $[W/m^2]$ . Some sample conditions are:

For roofs:  $\Delta R = 63 W/m^2$ , for walls:  $\Delta R = 0$ , for dark surfaces,  $\alpha/I_t = 0.052$ , which is the maximum value for any surface. To calculate heat transfer into the conditioned space,

$$Q_{solar-load} = U A (T_e - T_{in}),$$

where U is the overall heat transfer coefficient for the wall and A is the surface area for the wall. The term, U, includes convective and radiation effects by the internal and external airflow (see AHSHRAE's Fenestration Chapter for more details, [2]) and the wind outside, in addition to conduction through the walls.

All walls are insulated, say by a 1 inch layer of polyurethane of thermal conductivity of 0.026 W/m-K. Thus this value will be  $U \sim 1 W/m^2-K$ . The total solar load will then be around 500 W, and the total load that the cooling unit must dissipate will then be 2000 W ( $Q_{solar-load} + Q_{equipment}$ ). The area here refers to the 3 surfaces that can be illuminated simultaneously, with the roof always included.

### 4. Typical Cooling Systems for Enclosures

Outdoor enclosures may be cooled using a variety of methods

- Fully Active; Air-Conditioning/Refrigeration, Thermo-electric Coolers for Cooling
- Assisted (Semi-Active): Air-to-Air Heat Exchangers, Water-to-Air Heat Exchangers, Internal Fans, etc.
- Passive Means: Natural Convection

However, I want to focus on the use of PHOENICS to the thermal management of these outdoor enclosures. PHOENICS allows for the elucidation of flow fields and temperature distributions inside enclosures to determine if additional cooling is necessary (or additional fans, for example) to install more fans or higher capacity heat exchanger/air conditioner.

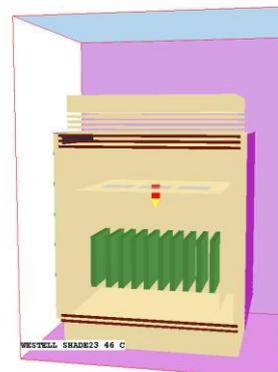


Figure 2 PHOENICS Model of Westell's SHADE23 Enclosure with Simulated Load (200 W)

Figure 2 shows a PHOENICS model of a fan cooled outdoor enclosure developed for Westell Technologies, Inc. of Aurora, IL—a customer of MJM Engineering Co. Here it is of note that the cooling system has an external cooling loop carried out by having air come in at the bottom of the side walls going up inside a channel (the side wall is a double wall) to the top induced by a main fan installed in the top double wall roof. Internally a 3-fan fan tray is used to induce an internal cooling flow to create uniform temperature conditions.

The simulation was carried out using PHOENICS' HOTBOX module. The model was steady-state, with various grid sizes depending on the number of fans and complexity of internal equipment, but, normally averaged over 5,000,000 elements. Radiation was not included in the model, the effect of solar loading was added on as a convective/conductive term.

As it can be seen from Figure 3, temperatures inside the electronics compartment hovers around 65 C (meeting the Telcordia requirement). Furthermore, the PHOENICS results indicate the temperature distribution inside the compartment along with flow distribution outside the enclosure, and in particular, the flow distribution internally to ascertain hot spots, especially when complex geometry electronic equipment is present.

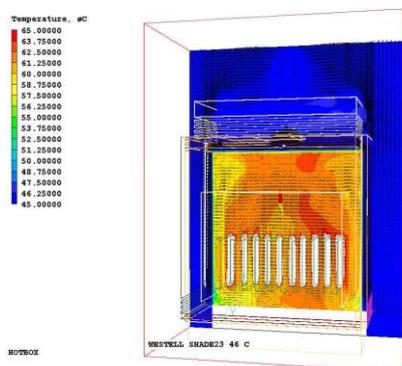


Figure 3 Results (Thermal and Velocity Fields) using PHOENICS of Westell's SHADE23 Enclosure with Simulated Load (200 W)

## 5. Conclusions

This article attempts to provide a brief overview of key topics in the thermal management of outdoor enclosures. It is believed that thermal performance can be determined and greatly improved through the use of PHOENICS.

## 6. References

1. McKay, J.R., "The Effect of Solar Radiation and Wind Speed on Air Temperature Rise in Outdoor Enclosures Containing Telephone Equipment," 10th International Telecommunication Energy Conference (INTELEC), San Diego, CA, USA, Nov. 1998
2. ASHRAE Handbook of Fundamentals, American Society of Heating, Refrigeration and Air-Conditioning Engineers, Atlanta, GA, 1981, 1986
3. Kreider, J.F. and Kreith, F., *Solar Energy Handbook*, McGraw-Hill Book Co., New York, 1981
4. Marongiu, M.J., "Some Issues In Experimental Testing & Methodologies In The Thermal Management Of Telecommunication Components, Systems & Enclosures," 17th International Telecommunication Energy Conference (INTELEC), The Hague, Holland, Oct 29-Nov. 1, 1995

MJM Engineering Co., 1163 E. Ogden Av., Suite 705-345, Naperville, IL 60563

Phone: +1 630/839-9489 Fax: +1 630/689-9447

e-mail: [maurice@mjm-engineering.com](mailto:maurice@mjm-engineering.com) <http://www.mjm-engineering.com>

### 3.2 Numerical Analysis of a Flame in a Flare Exposed to Crosswind by William Vicente, Martín Salinas, Engineering Institute, National University Autonomous of México, Ciudad Universitaria, Coyoacan, 04510, Mexico City

PHOENICS version: 3.5.1 PC-PENTIUM-D-WINDOWS

An important industrial safety application of PHOENICS is the prediction of radiation intensity contours in the vicinity of a flare. Simulation results of the turbulent combustion in a flare exposed to a crosswind are presented. The influence of the main cross-flow velocity on the flame aerodynamics is determined using a three dimensional domain in a Cartesian coordinates system. A fast-chemistry model with a one-step global irreversible reaction to simulate the formation of both CO<sub>2</sub> and H<sub>2</sub>O is used to represent the

combustion process. In addition, a radiation model is used to estimate the visible mean flame trajectory.

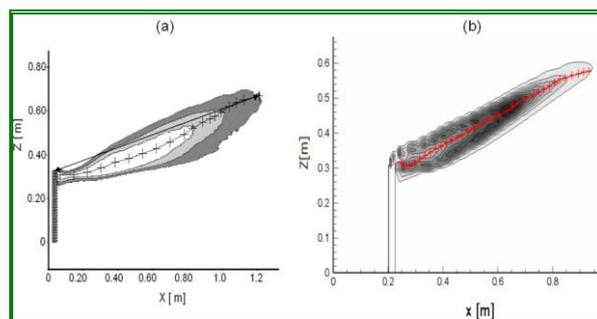


Figure 1. Mean flame trajectory for a cross flow velocity  $V_\infty$  of 2.0 m/s, ( $V_j/V_\infty=0.5$ ), a) Experimental data (Majeski, et al. 1999) and b) Numerical results (flame borders are defined by means of heat radiation contours).  $V_j$  is the propane discharge velocity and  $V_\infty$  corresponds to the cross flow velocity.

A propane jet that exits from a vertical cylindrical tube is placed in a perpendicular cross-flow air stream to perform the simulation. This air stream supplies the stoichiometric oxygen to develop the combustion process along the flare, for velocities ranging from 0.8 m/s to 12 m/s. A comparative analysis, with available experimental data from the technical literature, showed good agreement with the numerical results in terms of flame length and inclination angles.

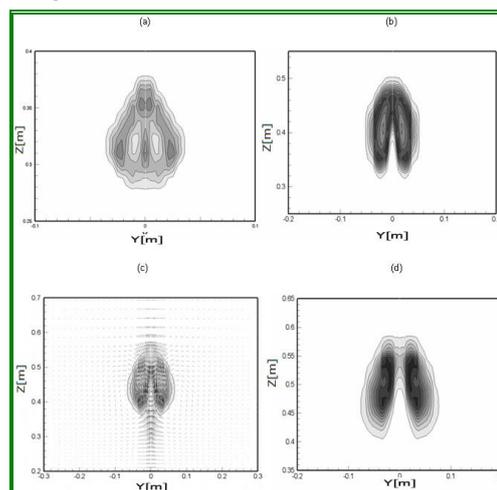


Figure 2. Mean shape for  $V_j/V_\infty=0.5$  at four different downstream locations of the stack: a)  $x/D=3.5$ , b)  $x/D=17$ , c)  $x/D=21$  y d)  $x/D=26$ . Flame borders are defined by means of heat radiation contours.  $V_j$  is the propane discharge velocity,  $V_\infty$  corresponds to the cross flow velocity and  $D$  is the diameter of the stack.

### 3.3 Numerical Analysis of Atmospheric Air Flow Entrainment in a House Stove Mixer

By William Vicente, Martín Salinas, Engineering Institute, National University Autonomous of México, Ciudad Universitaria, Coyoacan, 04510, Mexico City

PHOENICS version: 3.5.1, PC-PENTIUM-D-WINDOWS

PHOENICS is used to study the characteristics of atmospheric air flow entrainment which is generated by a fuel jet that is forced into a stove tubing assembly. The system configuration consists of a pipeline where the geometry of the inlet is varied in order to study its influence on the amount of entrainment air which enters the domain. The fuel jet is in contact with atmospheric air before entering the pipeline and the air-fuel mixture is produced inside the pipe as a consequence of the air entrainment. Comparisons with experimental data are made in terms of fuel concentration and entrainment air flow rate at pipeline outlet, where predictions showed good agreement.

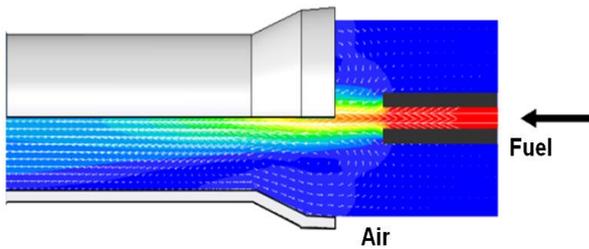


Figure 3. Numerical mean velocity fields at the pipe cross-section.

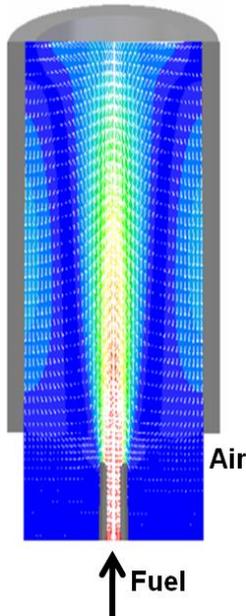


Figure 4. Flow field predictions of the fuel-air mixture.

#### 4) Consultancy Project Highlights by Paul Emmerson

##### 4.1 Data Centre Simulation for Thomson Reuters UK

A CFD model of a data centre was built, on behalf of Thomson Reuters, to analyze the cooling effectiveness of a proposed installation of high-density computer cabinets. Different cooling strategies were investigated with the objective of achieving near uniform operational temperatures across all cabinets.

The data centre employed a typical hot-and-cold-aisle ventilation strategy. Chilled air was supplied into the under-floor space via several CRAC (Computer Room Air Conditioning) units on each side of the room, feeding the cold aisles through perforated floor tiles. This ventilation air is heated as it cools the electronics in each cabinet, and then collects in hot aisles before returning to CRAC units, completing the cycle. Under-floor dampers are employed to regulate the flow through the perforated tiles. Redundancy of the CRAC units was taken into account, which meant less than 90% of the total cooling capacity was modelled.

In the simulation, the heat loading of server blades was averaged across each cabinet, with flow rates through cabinets defined by cooling fans. A specified amount of heat was extracted from the air as it passed through the CRAC units. Intake and exhausts for supply air were modelled, and a higher intake flow compared to extraction rate provided a positive pressure within the room, to minimise dust and warmer air from outside entering. Positive pressure was maintained, and mass flow within the domain balanced, using high resistance leakage paths through exit doorways. Major flow obstructions, such as cable trays under the suspended floor, power rails, lighting, beams and soundproofing near the

ceiling were all modelled. Also, additional heat sources were included, such as those due to fluorescent lights and external gains through walls, floors and ceilings.

The use of parameterised Q1 input files provided an efficient means of setting up multiple arrays of objects within the data centre. The final calculation mesh consisted of just over 10 million grid points. This provided sufficient refinement to resolve the geometry adequately and to capture the flow detail within the data centre. Two different types of K-ε turbulence model were investigated; the standard model and that with modifications of Chen and Kim. The standard K-ε model produced more stable convergence.

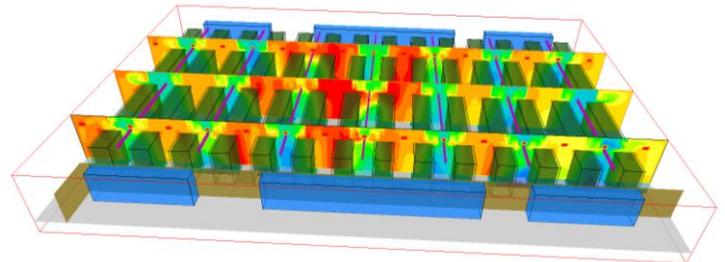


Figure 1 Temperature contours through computer cabinets (green) – under-floor and near-ceiling flow obstructions omitted

Steady-state simulations were performed for different cooling configurations. The average temperature of the hot return air and the chilled delivery air from each of the CRAC units was analysed to establish the system balance across the room. Any undesirable recirculation regions were identified; this is where warm air leaving the cabinets could get drawn back into a cabinet, before it has been re-chilled through the CRAC units. The use of cold aisle containment doors reduced the amount of recirculation. The presence of a technical corridor at one end of the room lead to some asymmetry in the predicted flow field, and had to be taken into account when optimising the cooling configuration.

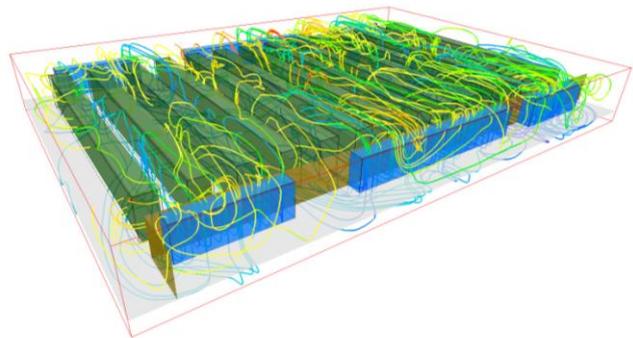


Figure 2 Streamlines through cabinets and return paths to CRAC units (blue)

**Academic Users are reminded that a condition of their Academic Licence is that they provide CHAM with reports (at least once annually) on the work which they undertake with PHOENICS so that this information can be included on the website and in Newsletters. Copies of these reports should be emailed to [cik@cham.co.uk](mailto:cik@cham.co.uk). Thank you.**

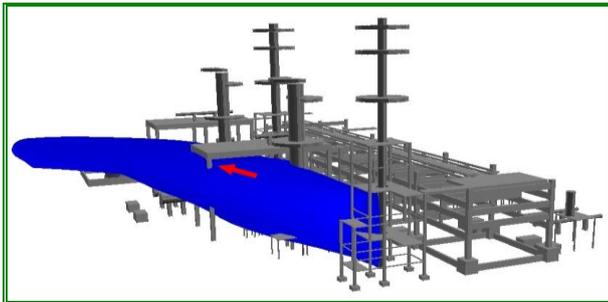
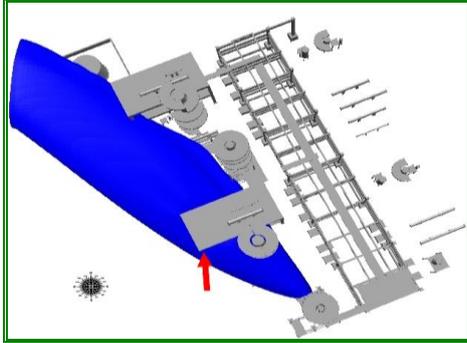
**All PHOENICS Users, and Agents, are invited to email articles for inclusion on the CHAM website and in the next Newsletter to [cik@cham.co.uk](mailto:cik@cham.co.uk). Thank you.**

**5) Agent Reports on Technical Work Done**

**5.1 Chemtech from Flavio Waltz**

**5.1.1 Allocation of flammable and toxic gases detectors (Petrobras REPAR):**

The purpose was to analyze the dispersion of toxic and flammable gases to determine the best position for detectors for these gases in 7 different Units of the Refinery. The evaluated scenarios were determined by a preliminary analysis of hazards and simulations were made using PHOENICS version 2008.



**Results: Detectors Allocated**

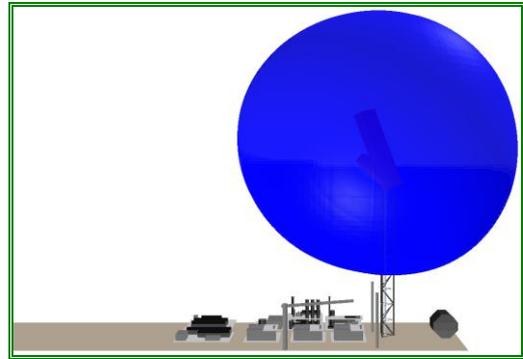
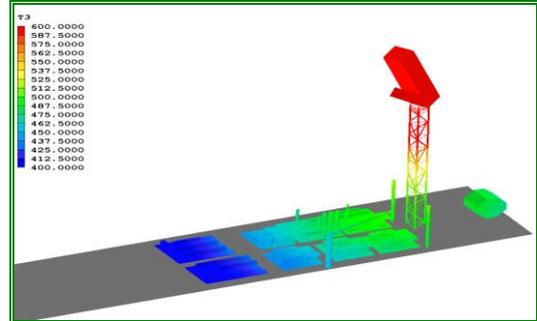


**5.2.2 Flare radiation and gas dispersion analysis (Petrobras P-58)**

Flare radiation levels and equipment temperature were calculated considering normal, and emergency, situations and using variations of flame emissivity and burner geometry. Simulations were performed using PHOENICS version 2008.

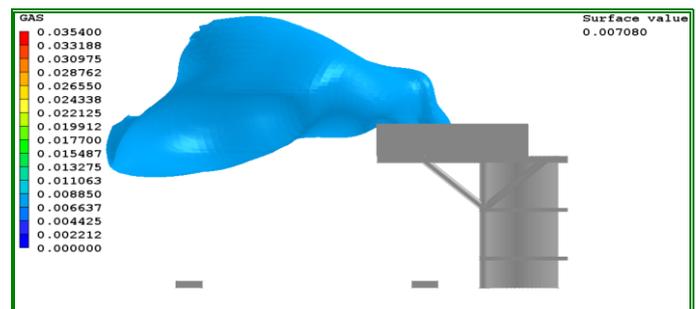
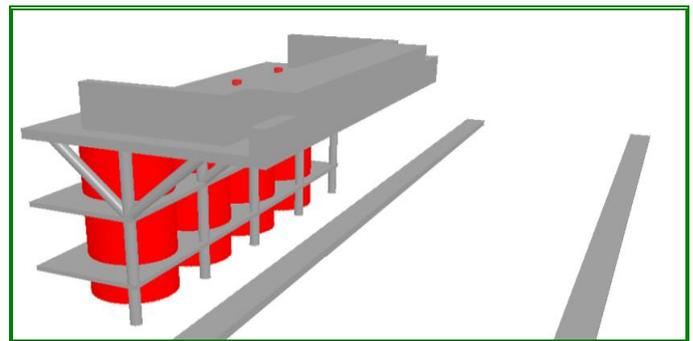
**Results:**

**Temperature**



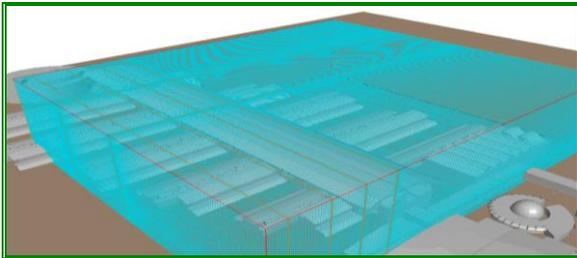
**5.2.3 Gas dispersion (Petrobras Revap):**

A leak on Petroleum Coke's reactor was analyzed to determine if the gas plume reached the treadmill bridge located at the Unit in a dangerous concentration (near or equal to the LFL – Lower Flammability Limit) considering five different wind conditions. PHOENICS version 2008 was used.

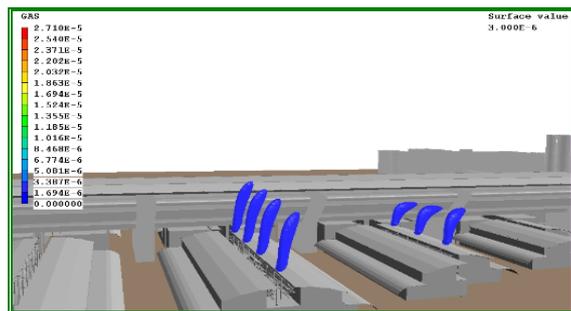
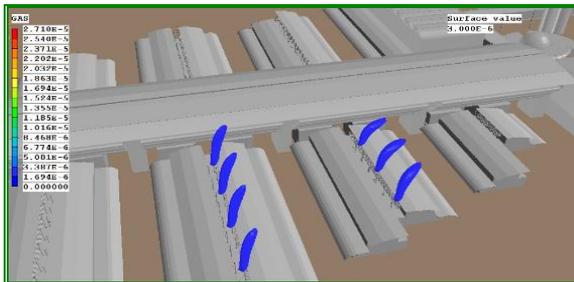


### 5.2.4 Chimney gas dispersion (Petrobras New Cenpes):

Gases emitted by the chimneys of New CENPES were analyzed to verify if the laboratories' exhaust system prevented the gases to reach the administrative facilities and laboratories in high concentrations. Simulations were performed using PHOENICS V2008.



Results:



### 5.2 ACFDA from Vladimir Agranat

Dr. Asish K. Sinha of U. S. Steel Corporation is using PHOENICS to study argon gas bubbling in steelmaking reactors. He has been able to model the mixing time of alloy additions in steel held in steelmaking ladles using argon gas rinsing from the tip of a vertical refractory lance immersed in the molten steel in the ladle using the IPSA model.

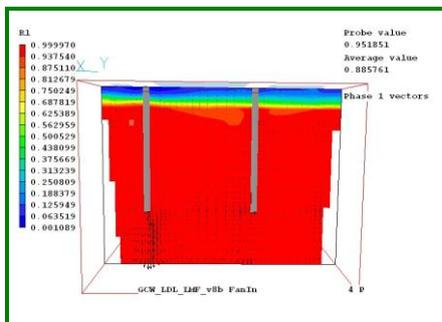


Figure 1: Distribution of volume fraction of molten steel in the ladle R1

## 6. News and Events 2009

### 6.1 Agent News and Staff (alphabetical order)

#### 6.1.1 ACADS ([www.ozemail.com.au/acadsbsg](http://www.ozemail.com.au/acadsbsg))

ACADS is CHAM's Agent in Australia, New Zealand and South East Asia. Their offices are located at 18 High Street, Glen Iris, Victoria 3146. Staff currently comprise Mr Murray Mason, Mr Trevor Kingston and Aiesha.



Murray Mason & Aiesha



Trevor Kingston

#### 6.1.2 ACFDA ([www.acfda.org](http://www.acfda.org))



Dr Vladimir Agranat (above) runs ACFDA (Applied Computational Fluid Dynamics) at 18 Rejane Crescent, Thornhill, Ontario L4J 5A5, Canada and sells PHOENICS products into the North American Market.

#### 6.1.3 ACT ([www.act-e.co.kr](http://www.act-e.co.kr))



Dr Dong-Seok Yeom (above) is the Managing Director of ACT 2000 (Advanced Computer Technology) based at Sesin Building, 290-1 Geoyeo-Dong, Songpa-Gy, Seoul 138-110, Korea and sells PHOENICS products into the Korean market.

