

## **RhinoCFD (Powered by PHOENICS)**

The “Earth” solver of CHAM’s PHOENICS CFD software forms the CFD *engine* of several third party products. The solver and its input menu structure have been, and are being, transposed to operate within the environment of specialist-application and multi-function third party CAD products. One such product is Rhino3D - a highly-successful 3D modelling tool produced by the McNeel Corporation—used worldwide and supplied and supported in the UK via Simply Rhino .

Rhino3D embodies a 3D working environment with tools allowing users to create CAD models of any shape, size or complexity and import third party CAD data for a range of file types. Its working environment is relatively easy to learn and is supported by training solutions designed to shorten the learning curve. Rhino3D also supports a range of plug-ins tailored for specialist applications.

CHAM has added to the range of popular plug-ins already supported by announcing a low-cost commercial variant of its powerful general-purpose CFD package, PHOENICS, called **RhinoCFD**. The product will shortly be available for download from the [www.Food4rhino.com](http://www.Food4rhino.com) website or via SimplyRhino, [www.simplyrhino.co.uk](http://www.simplyrhino.co.uk).

PHOENICS features activated in **RhinoCFD**:

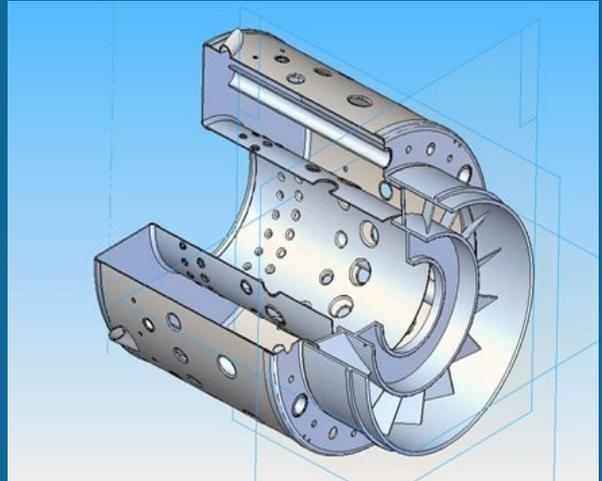
- PHOENICS-VR pre-processor (standard GUI);
- PHOENICS-PIL (PHOENICS Input Language);
- PHOENICS-2015 structured sequential CFD solver;
- VTK file handling and unstructured sequential CFD solver;
- PARSOL (Partial Solid) cut-cell geometry detection;
- Standard turbulence model options;
- DATMaker CAD and file conversion feature;
- Link to POLIS (PHOENICS Online Information System);
- VR output to Rhino display environment and PARAVIEW®.

PHOENICS features de-activated in **RhinoCFD**:

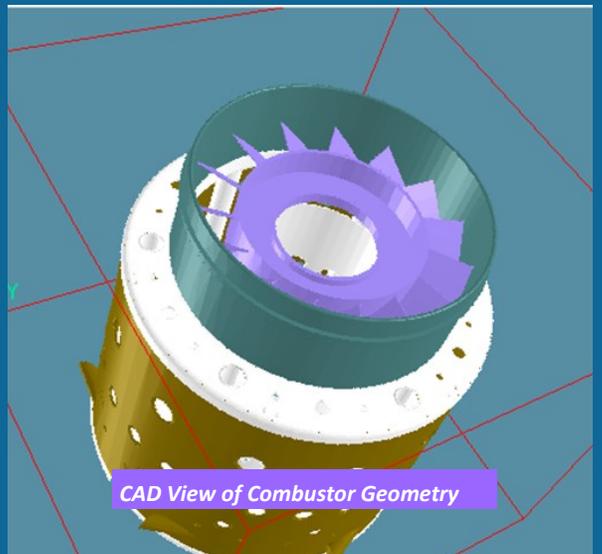
- Two-phase and multi-phase options;
- Multi-core and parallel-processing operation;
- All Special-Purpose Product variants (eg CVD, ESTER, FLAIR, HOTBOX, etc);
- Body-fitted coordinate meshes;
- PARSOL extensions (S-PARSOL and X-PARSOL);
- Non-standard turbulence models (eg Multi-Fluid-Model);
- Inbuilt POLIS (PHOENICS Online Information System);
- PHOENICS-VR post-processor.

**RhinoCFD** is limited to single-phase, sequential operation, running under Windows and offered at an attractive discounted price. Activation of special-purpose, applications-specific, variants is available at additional cost.

**RhinoCFD** offers annual or perpetual licensing terms with further price reductions for academic institutions.



*Combustor geometry displayed in Rhino3D and its PHOENICS-VR Plug-In*



*Contact [Sales@cham.co.uk](mailto:Sales@cham.co.uk) for further information and costings.*



# Natural Ventilation Modelling of an Apartment Using PHOENICS Flair

Tom Horsfield, CHAM

## Introduction

The PHOENICS environmental flow solver, FLAIR, is used to model natural ventilation inside a building. FLAIR allows the "mean age of air" to be calculated; which is particularly useful when identifying areas of poor ventilation.

In the case shown here, the air flow is driven by a pressure difference between opposite faces of the apartment, as might be seen in a high-rise building with open windows. The case demonstrates FLAIR's capabilities for an application defined by CHAM Agent, Shanghai Feiyi, and used for training purposes.

## CFD Model and Setup

As can be seen in Figure 1, the model shows an apartment with several open windows in its external walls. The geometry consists of two CAD files; one describing the exterior shape of the apartment and the other the internal geometry. Windows are specified using "ANGLED-OUT" objects where the external pressure is defined. Windows in the south- and east-facing walls have a higher relative pressure which is intended to drive air through the building to exit through north-facing windows.

A hybrid differencing scheme solves for all variables using a first-order upwind or central differencing scheme depending on cell Peclet number. The Chen-Kim modified k- $\epsilon$  model is employed to simulate turbulence. Mean age of air is calculated by solving for a passive scalar with a source term of  $1s^{-1}$ .

## Specifications

- Domain : 11.6m x 13.3m x 3.0m
- Relative pressure at south and east windows: 5Pa
- Relative pressure at north windows: 0Pa

## Mesh

For demonstration purposes the domain has been discretized using a uniform 60 x 60 x 30 Cartesian grid giving a uniform grid with cells 0.19m x 0.22m x 0.1m. See Figure 2.

## Results

Figure 3 shows pressure contours through the individual rooms of the building, at a height of 1.45m above the floor. The specified 5Pa over-pressure is applied on the external side of the south-facing and east-facing windows.

The pressure level in the rooms adjacent to these windows depends on the balances of upstream and downstream resistances for each room.

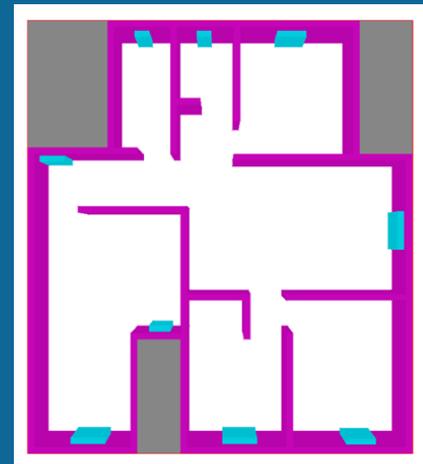


Figure 1: Geometry of Apartment

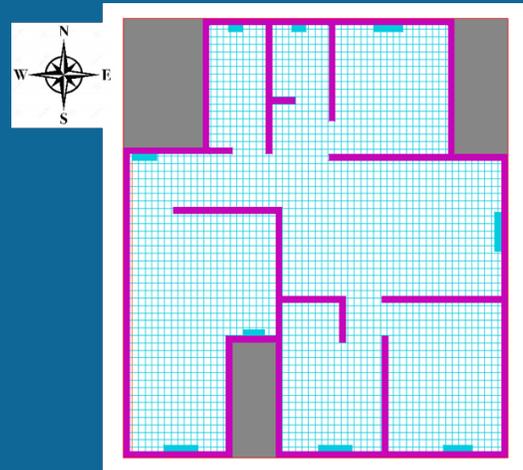


Figure 2: 60 x 60 x 30 Cartesian Mesh

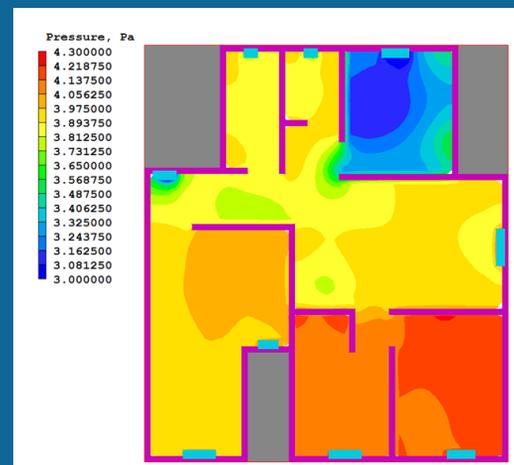


Figure 3: Pressure Contours at z = 1.45m



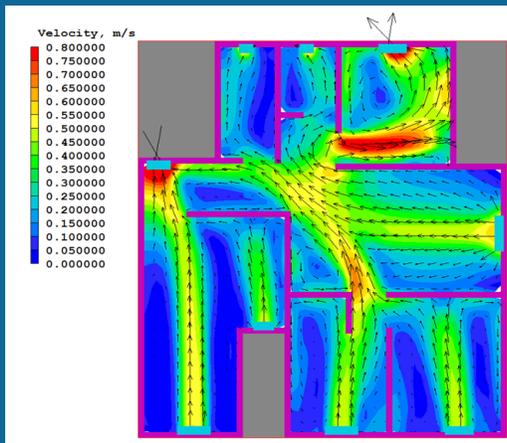


Figure 4 Velocity contours and vectors at z=1.45m

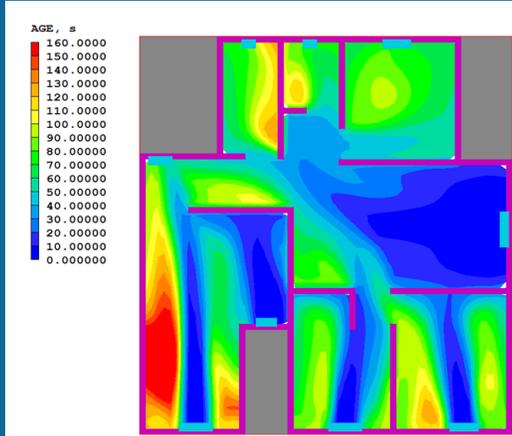


Figure 5 Mean Age of Air contours at z=1.45m

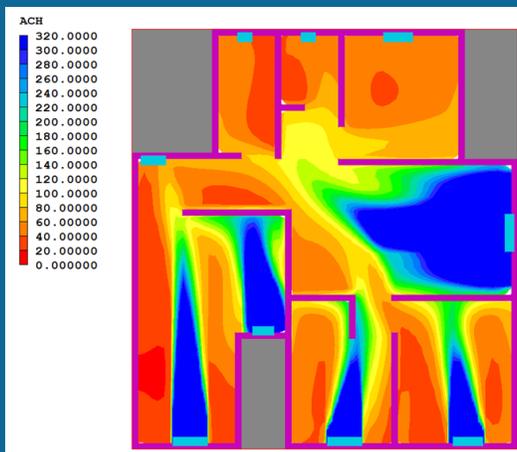


Figure 6 Air Changes Per Hour contours at z=1.45m

Figure 4 shows the ventilation pattern, at a height of 1.45m above the floor. Dark blue areas represent relatively poorly-ventilated corners.

Figure 5 shows the distribution of mean age of air, at the same height. The mean age increases as the air travels through the building, as expected. The highest values of mean age of air are in corners where the ventilation is relatively poor.

Figure 6 shows the mean number of Air Changes per Hour, again at a height of 1.45m above the floor. The highest numbers of air changes per hour occur where fresh air is constantly supplied and the lowest numbers are in poorly ventilated areas where the mean age of air is highest.

## Observations

The purpose of this training example is to show how PHOENICS FLAIR can simulate natural ventilation in an apartment. The pressure difference causes air to flow through the apartment from the south-and east-facing windows towards the windows in the north wall.

Age of air and air changes per hour are recorded to show the quality of ventilation across the apartment. These showed that areas with poor circulation can be generated when the internal design does not facilitate easy air paths throughout.

Use of PHOENICS FLAIR can highlight potential design deficiencies and thus prevent inattention to areas such as these causing problems for ventilation .

## News from CHAM



Professor Spalding will deliver a paper entitled “The Discretised Population Model of Turbulence” at an ERCOFTAC Conference (ETMM11) in Sicily in September (<http://www.ercoftac.org/events/etmm11/>).

### Abstract::

Turbulence influences fluid-dynamic phenomena in two ways:  
 1) By increasing the **fluxes** of momentum, energy and material resulting from time-average **gradients** of velocity, temperature and mass fraction; and

2) By increasing or diminishing the **sources per unit volume** of the same entities, as a consequence of rapid **fluctuations** of their values.

Both are of equal importance in practice; but the pioneers of present-day turbulence modelling (Boussinesq, Prandtl and Kolmogorov) paid attention only to the former; so their less-innovative followers have done likewise.

The present paper seeks to redress the balance, by describing an entirely different type of turbulence model; this concentrates on sources rather than fluxes; and therefore can simulate practically-important phenomena about which all popular turbulence models must necessarily remain silent.

It is here called the **Discretised Population Model**, its earlier-used name, *viz.* Multi-Fluid Model, having perhaps confused rather than enlightened. It can handle sources of all kinds, *e.g.* body-force, radiative, biochemical; but it is exemplified here by reference to sources associated with chemical reaction.

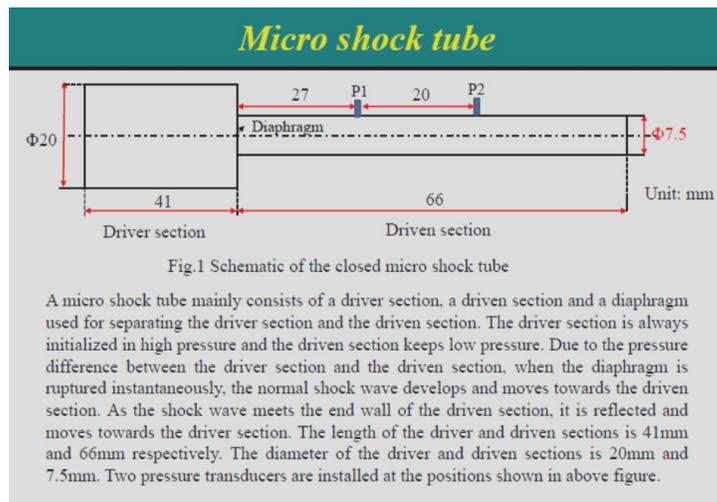


## CFD Modelling of a Micro Shock Tube

Dr Michael Malin & Mr Jason Cooke, CHAM

### Introduction:

Traditional macro shock tubes have a long history in aerodynamics, but more recently micro shock tubes have been used extensively in many engineering applications, such as for example micro-propulsion systems and drug-delivery devices for medical systems. A shock tube has closed ends, and the flow is generated by the rupture of a diaphragm separating a driver gas at high pressure from a driven gas at low pressure. This rupture results in the movement of a shock wave and contact discontinuity into the low-pressure gas, and an expansion wave into the high pressure gas. The major difference between micro and macro shock tubes is that the small flow dimension introduces additional flow physics. In particular, micro shock tubes experience shock attenuation from significant viscous effects at low Reynolds numbers. In addition, at high Knudsen numbers, there is slipping of the near-wall fluid due to non-continuum effects, and this acts to increase shock strength and aid shock wave propagation. The micro shock tube described below is one of several cases investigated by the Gas Dynamics Laboratory of the Andong National University of South Korea by employing a variety of well-known CFD packages for comparison purposes. CHAM was approached to provide guidance in using PHOENICS® to set up and simulate flow in the micro shock tube shown in Fig.1.



### Macro-shock tube:

As a precursor to considering the micro shock tube, CHAM undertook a 1D transient analysis of a traditional *macro* shock tube for which there is an analytical solution [1]. The purpose of this simulation was both validation and to demonstrate the ability of PHOENICS to capture the moving shock wave, contact discontinuity and rarefaction wave. These phenomena are also present in the *micro* shock tube, but as mentioned earlier, under the influence of significant viscous effects, and at high-Knudsen numbers, rarefaction effects also come into play.

The tube length and time period are chosen such that the computation ends before the two waves reflect from the ends of the tube. The simulation employs a uniform mesh of 100 cells to cover a 10m long tube with a cross-sectional area of 0.1m<sup>2</sup>. The transient simulation is run for a time period of 6.1 msec using 100 uniform time steps. The initial conditions are zero velocity with the following settings in each chamber: *Driver Gas*; pressure = 1.0 bar, temperature = 348.391 K and density = 1.0 kg/m<sup>3</sup>; *Driven Gas*; pressure = 0.1 bar, temperature = 278.13 K and density = 0.125 kg/m<sup>3</sup>. Wall friction is ignored and the energy equation is solved in terms of static enthalpy with an ideal-gas equation of state. The higher-order Van Leer MUSCL scheme is used for the discretisation of convection, and the default first-order Euler scheme is used for the transient terms.

CFD simulations were also made using the OpenFOAM® CFD solver set up via CHAM's recently-released "PH2OF" (PHOENICS to OpenFOAM) user interface. Both codes used the same computational grid, time step and spatial convection schemes.

Although time constraints precluded experimentation with the various numerical parameters that can influence the accuracy of the solutions, the comparisons made in Fig. 2 show good agreement between PHOENICS, OpenFOAM, and the analytical solution. The van-Leer MUSCL scheme was used in these simulations because the default hybrid scheme can produce inferior results due to numerical smearing. It is evident from Fig. 2 that OpenFOAM produces non-physical undershoots and overshoots in the predicted profiles.

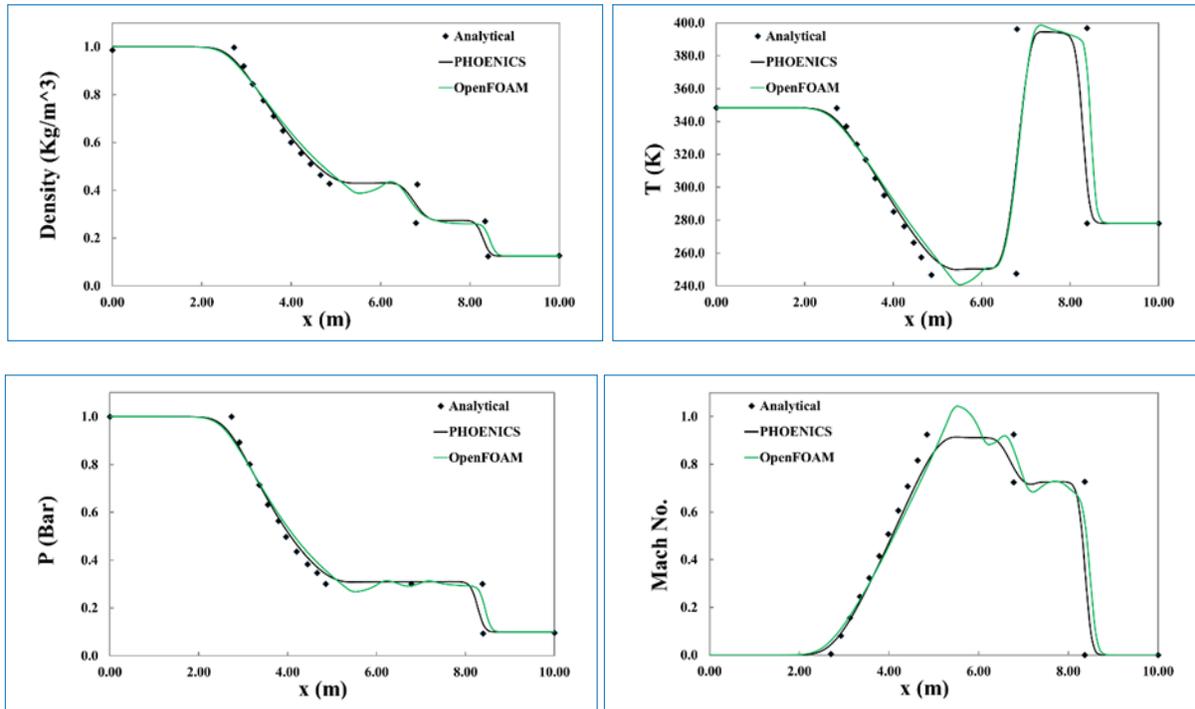


Fig. 2. Macro shock-tube: PHOENICS, OpenFoam, and analytical profiles at  $t = 6.31$  ms.

### Micro-shock tube:

In traditional shock tubes, the shock wave and the contact surface propagate essentially at a constant speed through the tube, but in micro shock tubes, a much thicker boundary layer develops behind the shock wave, which causes the contact surface to accelerate, the shock wave to decelerate, and the flow between these two waves to be non-uniform. For the micro shock tube shown in Fig.1, the initial conditions are zero velocity at a temperature of 300K with Driver and Driven chamber pressures of 9.0 and 1.0 atmospheres, respectively. In this case, the pressure of the driven gas is high enough for rarefaction effects to be absent. This situation has been investigated experimentally by Andong University, who performed static-pressure measurements over a time period of  $550\mu\text{sec}$ , as shown in Fig. 3. These measurements were made at the sensor points S1 and S2, which are synonymous with the locations P1 and P2 shown earlier in Fig.1. Fig. 3 also includes the CFD results obtained by Andong University using Fluent®.

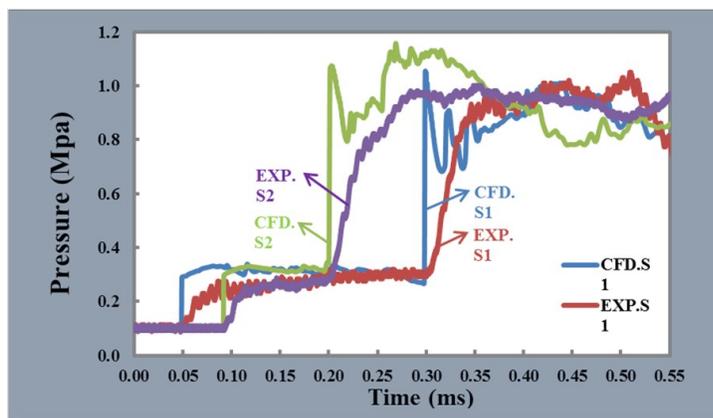


Fig.3 Micro shock-tube: Measured and predicted pressure histories at sensors P1 and P2.

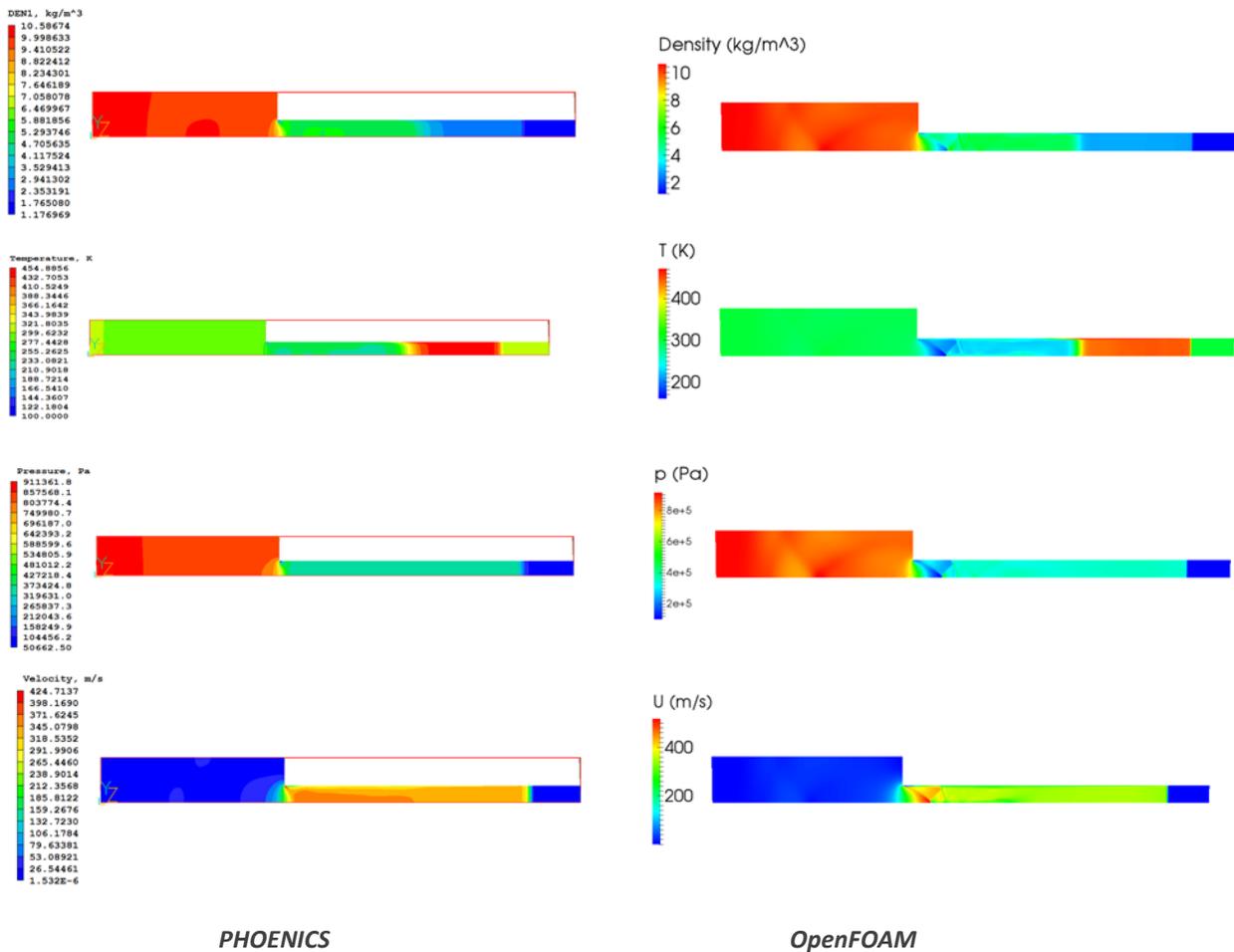
Transient, 2d axisymmetric simulations were made with all three codes using the structured, cylindrical-polar mesh specified by Andong University. This grid employs 160 radial by 1900 axial cells, of which 80 radial by 1172 axial cells are located in the driven section, and 160 radial by 728 axial cells in the driver section. Each simulation was run for a time period of  $500\mu\text{sec}$  using the uniform time steps given in Table 1. With OpenFOAM, convergence difficulties were encountered when using time steps larger than  $0.1\mu\text{s}$ . It is partly for this reason that PHOENICS ran 20% faster than OpenFOAM.

Table 1 also lists the main computational details, including the energy formulation and low-Reynolds-number turbulence model used by each CFD code. An additional run was made using OpenFOAM with the Jones-Launder low-Reynolds-number  $k-\epsilon$  model, but this made very little difference to the results.

For all simulations, Sutherland's law was used for the molecular viscosity together with a uniform specific heat of 1004 J/kgK, and laminar and turbulent Prandtl numbers of 0.71 and 0.86 respectively in the energy equation. The PHOENICS & OpenFoam simulations were run in sequential mode on a 3.4GHz Intel Core i7 CPU with 16GB RAM.

	<b>PHOENICS</b>	<b>OpenFOAM</b>	<b>Ansys Fluent</b>
Energy equation	Static enthalpy (h)	Internal energy (e)	Internal energy (e)
Time step	0.5 $\mu$ s	0.1 $\mu$ s	0.01 $\mu$ s
Velocity arrangement	Staggered	Co-located	Co-located
Turbulence model	2- layer $k-\epsilon$	$k-\omega$ SST	$k-\omega$ SST
Solution algorithm	Implicit SIMPLEST	Implicit PIMPLE (PISO/SIMPLE hybrid)	Density-based Coupled Solver with AUSM Flux Splitting
Time differencing scheme	1 <sup>st</sup> Order Euler	1 <sup>st</sup> Order Euler	2 <sup>nd</sup> Order
Convection discretisation scheme	MUSCL	MUSCL	Linear Upwind
Elapsed runtime (0.5ms)	22.3 hours	28.03 hours	Not reported

**Table 1: Micro Shock Tube: Main Computational Details**

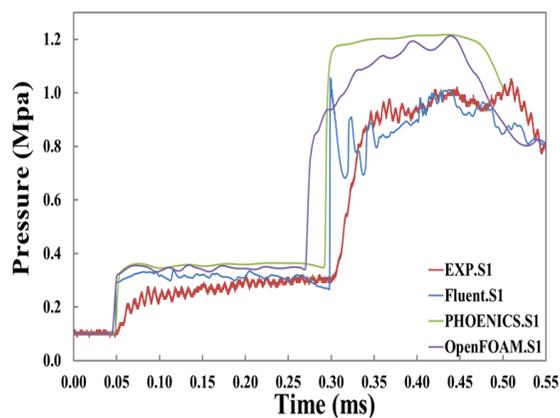


**Fig.4 Micro shock-tube: Contour plots of field variables after 0.1ms.**

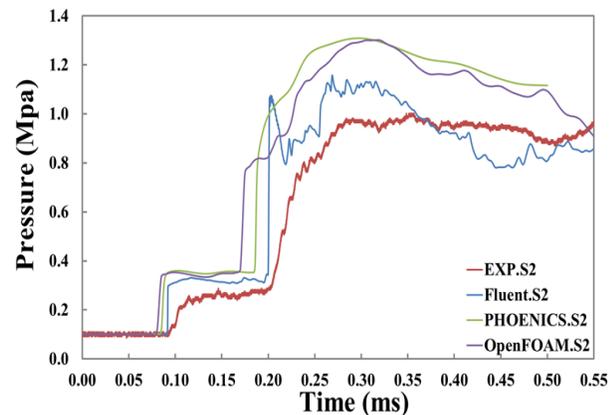
Fig. 4 compares the contour plots of density, temperature, pressure and absolute velocity produced by PHOENICS and OpenFOAM some 0.1msec after rupture of the diaphragm. The plots show the normal shock wave moving to the right through the driven section, and the expansion wave propagating to the left through the driver section. It can be seen that the shock wave gives the driven gas a severe acceleration accompanied by a jump of temperature, pressure and density. It is not shown here, but the predictions also show the expected further increase of temperature, pressure and density of the driven gas when the shock wave is reflected back from the closed end wall.

Fig. 4 also reveals that OpenFOAM predicts sharper rarefaction waves expanding into the driver section, as well as sharper waves and reflections just downstream of the diameter reduction in the driven section. The crisper wave capture of OpenFOAM is probably due to the much smaller turbulent viscosities produced by the  $k-\omega$  SST model. Specifically, the 2-layer  $k-\epsilon$  model introduces a smearing effect by producing excessive turbulent viscosities across shock waves and contact discontinuities. It would be fairly straightforward to implement the  $k-\omega$  SST model in PHOENICS by means of the InForm facility in the Q1 input file. An additional PHOENICS run was made using the Sarkar compressibility corrections [3] to the 2-layer  $k-\epsilon$  model, which are intended to reduce the predicted turbulence levels due to dilatational effects in high-speed compressible flow. This modification effected very little change in the solution.

Figures 5 and 6 compare the measured and predicted pressure histories at the two sensor points located on the outer wall of the driven section. A comparison is made between PHOENICS, OpenFOAM, Fluent and the experimental data. The results produced by all three codes are broadly in line with experiment. PHOENICS and Fluent predict well the shockwave arrival times at S1 and S2 for both the initial and reflected shock wave, respectively. However, OpenFOAM predicts a faster arrival time for the reflected shockwave.



**Fig. 5 Micro shock-tube: Measured and Predicted pressure histories at the Sensor S1.**



**Fig.6 Micro shock-tube: Measured and Predicted pressure histories at the Sensor S2.**

Both OpenFOAM and PHOENICS over-predict the pressure level within the driven section of the shock tube during the latter stages of the simulation. It is evident that Fluent was able to predict a better level for the pressure in this period, perhaps due to the much smaller time step combined with second-order time differencing. However, the large fluctuations in pressure predicted by Fluent are not present in the measurements, and they may indicate the presence of non-physical undershoots and overshoots in the solution due to unbounded nature of the linear upwind scheme. Such non-physical niggles were also observed by Lamnaouer [2] in his simulation of macro shock tubes, which he attributed to the absence of a flux limiter in the MUSCL scheme of Fluent.

Further work is needed to investigate the reasons for the discrepancies between the three codes and experiment.

## References

- 1) Sod, G. A. (1978), "A survey of several finite difference methods for systems of nonlinear hyperbolic conservation", J. Comp. Physics, Vol. 27, pp. 1-31.
- 2) Lamnaouer, M., (2010), "Numerical modelling of the shock tube flow fields before and during Ignition delay time experiments at practical conditions". PhD Thesis, University of Central Florida, Orlando, Florida, USA.
- 3) Sarkar, S., Erlebacher, G., Hussaini, M. Y., and Kreiss, H. O., (1991), "The analysis and modelling of dilatational terms in compressible turbulence", J. Fluid Mech., Vol. 227, pp 473-493.

® PHOENICS is the registered trademark of Concentration Heat And Momentum Limited [CHAM]

® Fluent is the registered trademark of Ansys Incorporated

® OpenFOAM is the registered trademark of OpenCFD Limited

## S & C Thermofluids – The Academy

Over the summer months in 2016, S & C Thermofluids is hosting a Training Academy for undergraduate and graduate students wishing to gain practical experience in CAD, CFD and aerospace engineering.

Students will receive a full programme of training from S&C engineers and consultants in a variety of meshing and fluid dynamics software packages, including the company's in house PLUMES code which is supported by the PHOENICS solver.

S & C would like to extend its thanks to CHAM for providing temporary licenses for PHOENICS to the Academy, allowing the students not only to gain confidence in applying their theoretical and academic knowledge to a commercial program but actively to participate in real world CFD projects and present unique work to high level companies such as DSTL.

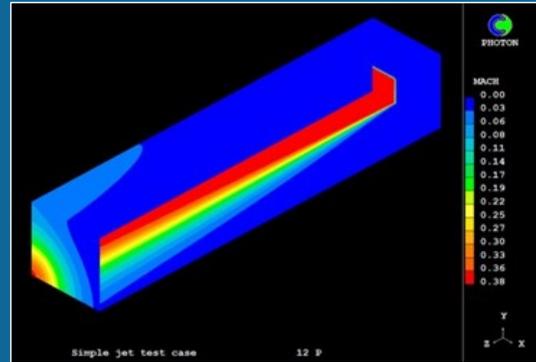
They will then have the opportunity to refine and practice their new skills by assisting staff on innovative Research & Development projects, including concept design, software development and code validation.

The company's vision is to provide a unique and vibrant working environment to build an inspiring platform for the engineers of tomorrow. The Academy opened in 2015, when four students were employed from June to September and completed an impressive number of research and development tasks, including liaising directly with CHAM to explore the capabilities of parallel PHOENICS on a High Performance Cluster at the Centre for Modelling and Simulation (CFMS) at the Bath and Bristol Science Park. The feedback from the scheme was very positive and S & C Thermofluids is delighted to be able to offer this opportunity once again this year.

***“My time at the academy increased my problem solving and organisational skills. Additionally I got the chance to present to industry experts which was a unique experience.”***

***“In a world where one needs previous work experience in order to get a job, I think the S&C Academy is an amazing opportunity for anyone that takes part.”***

S & C Thermofluids Ltd is an experienced body of professional engineers, specialising in fluid dynamics and thermodynamics. Founded in 1987, the company delivers low cost, high quality engineering solutions for products, processes and systems over many disciplines, including civil and defence aerospace. These solutions are an innovative combination of computational prediction and experimental tools, providing the most powerful and systematic route to product development.



2015 Academy group – Rory Davis, Sam Thomas, Oliver Sutcliffe and Ryan Dyer

For more information on S & C Thermofluids please visit: [www.thermofluids.co.uk](http://www.thermofluids.co.uk)



## Modelling of pressurization in HVAC systems

Marek Magdziarz, Wentylacja Strumieniowa (Poland)  
Agnieszka Belz, Norklima (Poland, Norway)

There is often a need to pressurize buildings, or areas within buildings, in order to protect their occupants from process emissions from outside, or to retain a high level of air purity inside. Examples include refineries, the chemical processing industry, electronics production, hospital environments, and so forth. To ensure that HVAC systems carry out this function effectively, HVAC designers can now accurately test and verify their designs with the aid of CFD technology.

The function and construction of the building described below is unique and is located within a refinery. Its function is technical supervision offices and to house and protect people in the event of pollution release outside (mostly H<sub>2</sub>S). It was decided to check the technical assumptions and functional design of the HVAC system operating in various modes – operating scenarios – using CFD modelling.

The aim of the project, and the HVAC system, is to create and retain an overall overpressure in the interior spaces of the building (see 3D model shown in Figs. 1 and 2) relative to the external pressure, wherein the pressurization system was adapted for different room functions. Additionally, it was required that the HVAC control system had to sustain four main operating conditions providing different overpressure values and ventilation air volumes, which were to be verified by four CFD analyses, as follows:-

Scenario no 1: HVAC and control system work under normal operation in the *summer time*, with no fire or pollution hazard outside the office block. During this scenario, the following were considered: heat gains (from humans, lights, equipment, external walls/floor/roof), air supplied with constant air temperature chilled and controlled by the AHU (Air Handling Unit), the individual DX-multi-split air conditioners (with heat pump option) dedicated for server room, electrical-power supply room, battery room and 3 office rooms.

Scenario no 2: HVAC and control system work with normal operation in *winter time*, with no fire or pollution hazard outside the office block. During this scenario, the following were considered: heat losses, air supplied with constant air temperature heated and controlled by the AHU, the individual DX-multi-split air conditioners with heat pump operation dedicated to individual rooms.

Scenario no 3: HVAC and control system work with emergency operation with *partial air recirculation* (in both winter & summer time), with pollutants having been detected in the fresh air ductwork of the AHU. The rate of air entrainment is reduced to a minimum, and the air is filtered.

Scenario no 4: HVAC and control system work with emergency operation with *full air recirculation* (in both winter & summer time), when the filtration system is not able to clean the air.

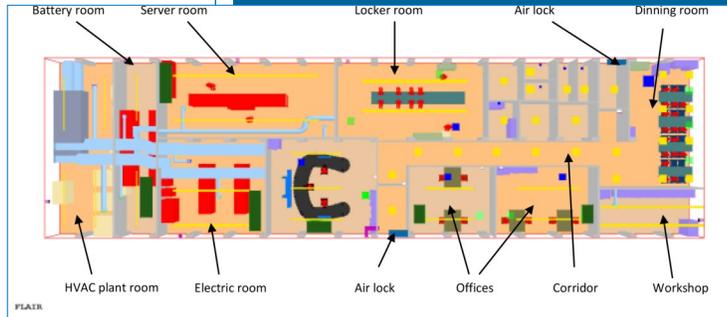


Fig 1: Top View of 3D Model



Fig 2: Side View of 3D Model



The design involves the use automatic pressure transfer dampers below the suspended ceiling in the office rooms. The air is transferred towards the corridor if the pressure rises above a set value +100Pa and the pressure in the corridor has a lower value. The air is transferred from the corridor towards air locks through the pressure transfer dampers, if the pressure in the air lock has lower value than in the corridor. Finally, the pressure in air locks is released outside of the building through pressure relief dampers, fitted on external walls and above the floor, set on value +50Pa of overpressure.

In the technical rooms, pressurized air is transferred similarly towards the outside of the building through automatic pressure relief dampers. There are fire dampers and gas-shut-off dampers - located near the pressure transfer dampers - that control flows and adjust the system according to its operating mode (viz; *normal mode* using fresh air from outside, or *1<sup>st</sup> emergency mode* with partial air recirculation, or finally *2<sup>nd</sup> emergency mode* with full air recirculation).

The HVAC system in Scenarios 1 & 2 operates using fresh air from outside and the objective of the ventilation scheme is to produce pressurization (overpressure) in the different areas shown in the CFD model (see Figs.1 & 2):

- HVAC plant room, server room, electrical (power supply) room, lockers rooms with toilets, corridor with dining room, 3 offices: +100Pa
- 2 air locks, battery room, workshop: +50Pa

In Fig. 3 you can see result of the CFD simulation showing the pressure distribution at a time of 1 second for Scenario 1; this assumes that the volume of air supplied and extracted start at the design value. Fig. 4 shows the pressure ranges after 11 sec.

The CFD simulation also predicted the temperature and air flow distribution from diffusers and DX split conditioners during cooling mode in summer time – Scenario 1 (e.g. office room – Figs. 5 & 6) and during heating mode in winter time –Scenario 2 (e.g. temperature distribution above the floor – Fig. 7).

The HVAC system in Scenario 3 acts with partial outside (fresh) air and the objective is to produce pressurization (overpressure) in various spaces:

- HVAC plant room, server room, electrical (power supply) room, lockers rooms with toilets, corridor with dining room, 3 offices: +100Pa
- 2 air locks, battery room: +50Pa
- workshop: not pressurized.

Fig. 8 (on page 12) shows the result of the CFD simulation showing the pressure range at a time of 1 sec. for Scenario 3. As before, this assumes that the volume of air supplied and extracted commence at the design value. Fig. 9 shows the pressure range after 11 sec.



Fig 3: Pressure Ranges in Rooms after 1 sec., in summer mode— Scenario 1

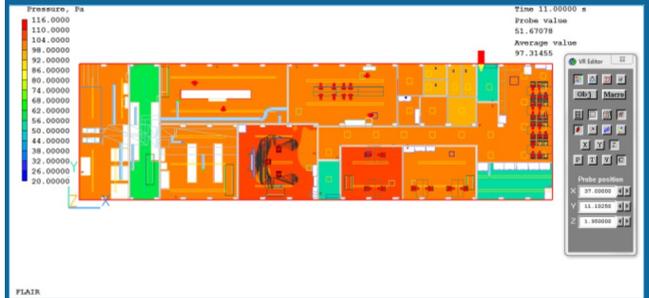


Fig 4: Plot of Pressure Ranges in Rooms after 11 sec., in summer mode—Scenario 1

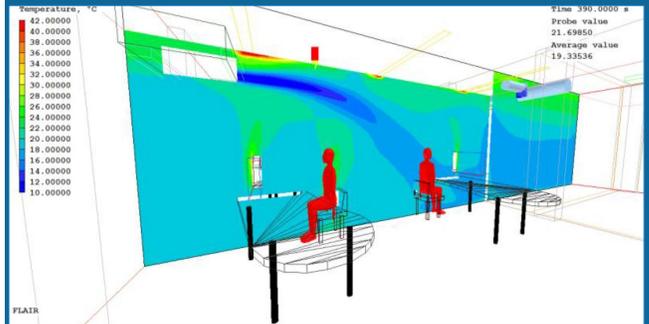


Fig 5: Plot of Temperature Ranges in Office/Monitoring Room, in summer time—Scenario 1

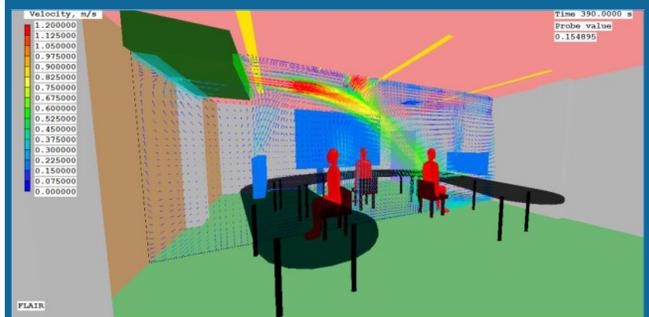


Fig 6: Plot of Velocity Vectors with DX-split air conditioner in Office/Monitoring Room, in summer mode—Scenario 1

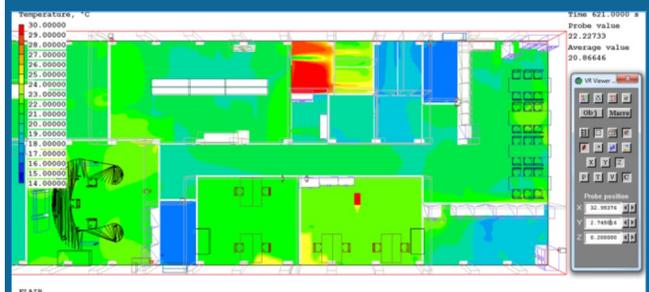


Fig 7: Plot of Temperature Ranges Above the Floor, in winter time— Scenario 2

The analysis of each scenario simulated showed that the pressure in the rooms attained the required level very quickly and no case exceeded 9 seconds. The rate of increase in pressure can also be calculated using mathematical formulas, not only via CFD modelling.

The HVAC system in Scenario 4 operates with the full recirculation of air (with the resulting lack of oxygen compensated for by a “life-support system”) and the objective of the ventilation system is to produce pressurization (overpressure) in various spaces:

- locker rooms with toilets, corridor with dining room, 3 offices: +100Pa
- 2 air locks: +50Pa
- HVAC plant room, server room, electrical (power supply) room, battery room, workshop: not pressurized

The biggest challenge for any pressurized ventilation system is the correct design of controls and the system of automatic valves regulating the pressure, as well as checking the air-tightness of doors and windows, and the proper estimate of quantities, parameters and identifying the location of potential leaks within the facility.

In fact, it may sometimes be necessary to operate the facility as a completely-sealed entity, where the doors and windows and installation culverts are sealed carefully to the exclusion of any natural ventilation. In some cases, there may be a necessity to design the exterior walls and ventilation ducts, fire dampers and valves to close in a gas-tight manner to protect the building against penetration of dirt or fumes during periods when the premises are not pressurized.

As it was a pioneering project using CFD discipline for this type of application, CHAM’s Dr John Ludwig created especially dedicated models of pressure transfer and pressure relief dampers with mass, temperature and momentum transfers.

The performing of such calculations, estimating and verifying ventilation requirements is a difficult and responsible task, as made clear by the requirements of Polish Building regulations. These state that only people with appropriate education and professional experience can perform such calculations.

The same applies to the CFD calculations where the scope of the calculations has to satisfy technical and physical goals. If such calculations are performed by people without suitable building expertise, the result may be the refusal of insurance companies to pay compensation in the case of technical problems with the installation.

Email: [marek.magdziarz@wentylacja-strumieniowa.com.pl](mailto:marek.magdziarz@wentylacja-strumieniowa.com.pl)



Fig 8: Plot of Pressure Ranges in Rooms after 1 sec., in emergency mode—Scenario 3



Fig 9: Plot of Pressure Ranges in Rooms after 11 sec., in emergency mode—Scenario 3



Fig 10: Plot of Velocity Plans in Cross-Section through Pressure Transfer Damper in wall between air lock and corridor—Scenario 4

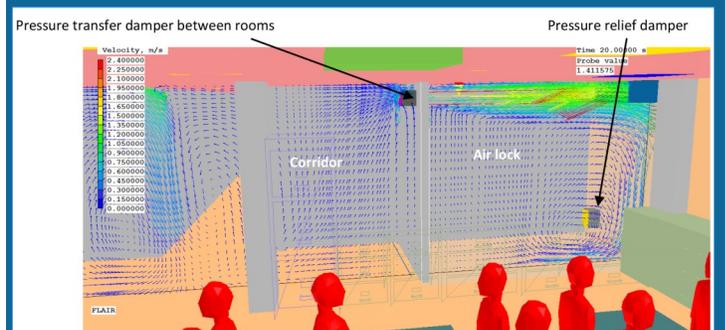


Fig 10: Plot of Velocity Plans in Cross-Section through Pressure Transfer Damper in wall between air lock and corridor—Scenario 4

Contents	Pg
PHOENICS 4 RHINO 3D: CHAM Product Update	2
Natural Ventilation Modelling of an Apartment Using PHOENICS-FLAIR	3
CFD Model of a Micro Shock Tube	5
S & C Thermofluids—The Academy	9
Modelling of Pressurization in HVAC Systems	10