PHOENICS Newsletter

CHAM

This issue

Concentration, Heat and Momentum Limited (CHAM)
Bakery House, 40 High Street, Wimbledon Village, London, SW19 5AU, England
Tel: +44 (0)20 8947 7651 Email: phoenics@cham.co.uk Web: www.cham.co.uk

Autumn 2020

Dear Reader

We are nearing the end of this most strange of years when what we took for granted became unsure. We, as with much of the world, were asked to lock down, work from home, stay indoors and generally isolate ourselves. We locked down end March and were allowed back to our work place in Wimbledon Village start July.

It was a relief to return. Working remotely was a reasonable plan B but does not enable technical, and other, important discussions and interactions made possible by being together. We returned and, slowly, life began to look almost normal.

August and September passed with people, in the main, being careful, choosing to holiday here rather than abroad, meeting in small groups and, whilst not relaxing, being perhaps more hopeful. Children were allowed to return to school. University terms started. The death rate slowed. The wider availability of tests meant that, unsurprisingly, case numbers increased but, fortunately, hospitalizations did not. Progress might be slow but it seemed to be happening.

Now it is November and it seems as if we, as a country, are going backwards.

We are, again, locked down with "effective" working from home being encouraged. Because the company is not as effective when working remotely as when on site, CHAM is remaining open and staff are able to come in 2 days per week in 2 separate teams. We have health & safety measures in place so in-house risk to our staff is minimized. It continues to be good to work within our professional and business environment. It seems optimistic, however, to believe that we will be able to return, as a team, for an entire week under the one roof until the New Year.

We have produced two videos relating to the virus and its spread. One, using PHOENICS, shows two people seated across a table in a restaurant. One is shown breathing. This can be seen at ……; there is further information on page 8. The second, using RhinoCFD (powered by PHOENICS) shows the spread of a sneeze on a tube and can be seen at……. Work is also being undertaken for others relating to modelling virus spread.

CHAM is still unlocking PHOENICS for use at home. To avail yourself of this facility please contact phoenics@cham.co.uk.

To include a technical article, or item of news, in the Winter Newsletter please send, as word documents, to news@cham.co.uk.

We hope all reading this Newsletter - Clients, Agents, Friends made over the years - continue to be healthy. We look forward to hearing from you and will continue to work to provide our consulting, technical, or other, services.

Kind Regards

Colleen Spalding
Managing Director

Contact US

Should you require any further information regarding our offered products or services please give us a call on +44 (20) 89477651; alternatively you can email us on sales@cham.co.uk.

Visit our website at www.cham.co.uk; we are also on the following social media sites:

PHOENICS Modelling of a Self-Propelled Body in Stratified Water, by Dr R P Hornby.

Introduction

A previous article in the PHOENICS Newsletter (Ref 1) has shown how the MOFOR concept can be adapted to allow modelling of a moving body acted on by specified forces. This provides a means of investigating more realistically aspects of the dead water effect applied to vessels moving in stratified water. The starting point has been the experiments carried out at Delft University by Professor Leo Maas (Ref 2) where rectangular and toy boat models were towed in a 2m length stratified tank. These experiments illustrated the significant effects on boat speed of enhanced drag due to generation of internal waves below the surface.

In this article, PHOENICS is used with the modified MOFOR capability to model the motion of a rectangular body with given thrust forces in a slightly modified version of the experimental tank. The tank modelled is 4*m* in length, 0.25*m* wide and 0.1525*m* deep and is filled with water of density 1027 kgm^{-3} to a depth of 0.1m and water of density 1000kgm^{-3} above that to a height of 0.1525*m*. The rectangular body is 0.14*m* in length, 0.1*m* wide and 0.03*m* deep with the top face level with the surface so that the body weight (0.42*kg*) is calculated as the body volume multiplied by the upper water density.

The tank is modelled using 400, 13, 16 cells uniformly distributed in the axial, transverse and vertical directions. Surface friction is not taken into account as its effect is considered to be small and the flow is assumed to be laminar. The thrust forces are chosen to give body speeds that are similar to the internal wave speed associated with the given stratification (which is approximately 0.1ms^{-1}, but slightly less in the simulation due to the relative coarseness of the grid used). The time step was 0.05*s* and the simulation period for each case was 50*s*. This time step ensured a Courant Number less than one in all simulations.

Results

The first five simulations used a constant thrust force but varied it from the maximum to 60% of the maximum. In each case the body starts from rest.

The results are shown for the body speed as a function of time in figure 1. It can be seen that there are significant effects caused by the stratification. In all cases there is a strong effect due to increased drag as the body speed approaches the internal wave speed after about 5s. At full thrust the body is able to recover and the body speed increases but at 90%, 83%, 75% and 60% of maximum thrust the speed reduces and gradually levels out. The effect is obviously sensitive to the level of thrust as a relatively small reduction in thrust (from the maximum to 90% of the maximum) leads to a dramatic reduction in body speed (from 0.14ms-1 to 0.05ms-1). There are then less dramatic reductions in speed between the 90% full thrust value and the 83%, 75% and 60% values.

An additional four simulations were performed to further illustrate the dramatic effect of the stratification (figure 2). In the first simulation the thrust was set at 60% of the maximum and the water density was set at 1000kgm-3 in the whole calculation domain. So this particular run shows the body speed attained in the absence of stratification (0.082ms-1). The second run is the same as the first except that the domain is stratified as described above. In this case the speed attained is significantly reduced (0.045ms-1). The third run is as for the second, but at 15s the thrust is increased to 83% of the maximum value. The fourth run is as for the third except that at 15s the thrust is ramped up to its maximum value. The latter two cases show that even these relatively large increases in thrust do not produce the expected response in speed (especially for the 100% ramp up case, see figure 1). Another feature of the latter two cases is the oscillatory nature of the speed response. This is also apparent in the uniform thrust cases shown in figure 1.

If the body is stopped by zeroing the thrust then the simulations show the wave train created by the initial motion overtaking the body. The oscillations in speed occur when the thrust is not sufficient to overcome the increased wave drag, so the body slows down, wave drag reduces enabling the body to increase speed and the cycle repeats.

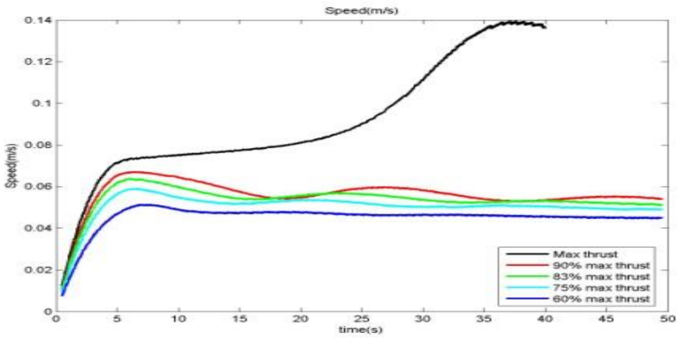


Figure 1. Body speeds at various thrust values as a function of time

centerline and tank wall at 40s from start for the case at 60% thrust changing to 100% thrust after 15s are shown in figure 3. Figure 4 gives a clearer idea of the waves created using vertical velocity contours and flow vectors. This figure clearly shows mode 1 internal waves travelling both behind and in advance of the moving body (the position of which can be identified by the area of parallel surface velocity vectors).

Three further pairs of runs were carried out over a reduced time span of 25s to assess the effect of the body draft (effectively the body thickness in these simulations). The first run of each pair was non-stratified, (for reference), the second was stratified.

Figure 5 shows the results for body speed for body drafts of 0.015m, 0.03m and 0.04m corresponding to 29%, 57% and 76% of the top layer depth. For comparison, the equivalent results for the non-stratified cases are also given. It can be seen that a significant reduction in speed is encountered for all the stratified cases.

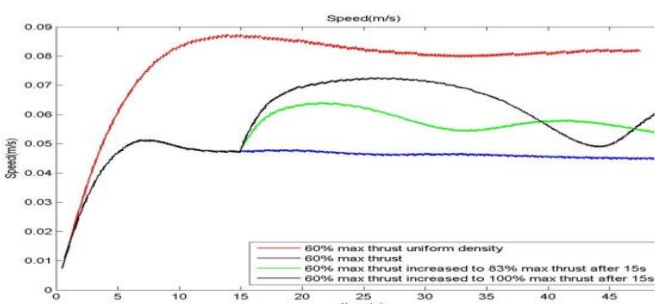
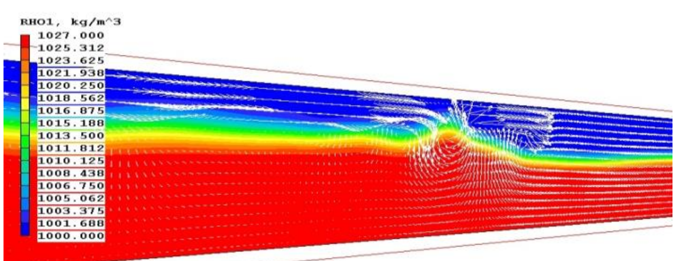


Figure 2. Body speeds at initially 60% of maximum thrust with ramping of power to 83% and 100% of maximum thrust after 15s.



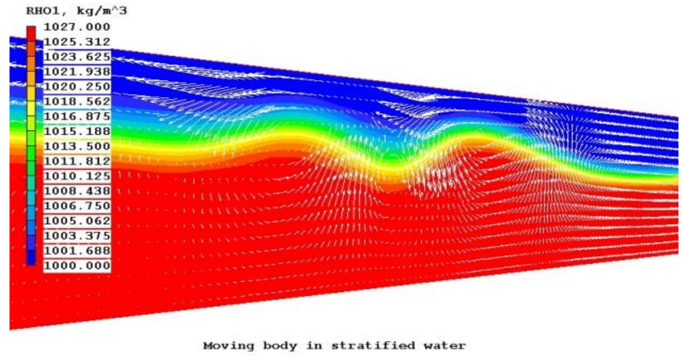


Figure 3. Density contours and velocity vectors after 40s for the case starting at 60% thrust, increasing to 100% thrust after 15s. Vertical scaled by factor of two for clarity. Left: Density contours and flow vectors along the tank centerline, Right: density contours and velocity vectors along the tank wall.

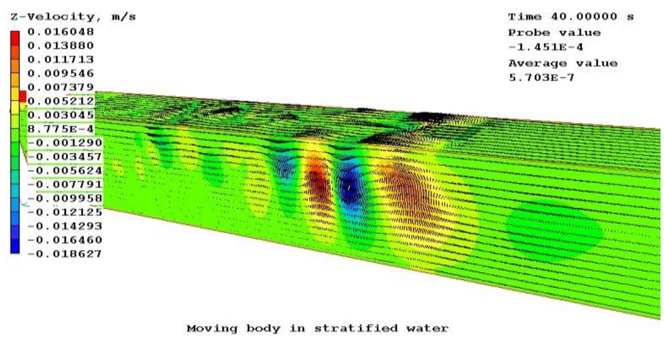


Figure 4. Surface vertical velocity contours and velocity vectors after 40s for the case starting at 60% thrust, increasing to 100% thrust after 15s.

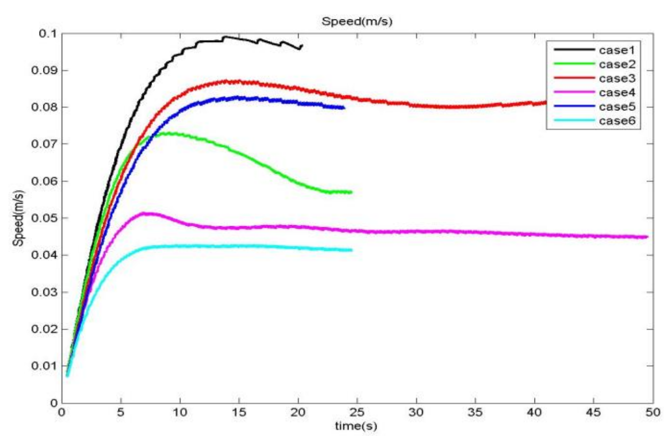


Figure 5. Body speed versus time for 6 simulations. Cases 1 and 2 have body draft of 0.015m with case 1 being the non-stratified equivalent to case 2. Cases 3 and 4 have body draft of 0.030m with case 3 being the non-stratified equivalent to case 4. Cases 5 and 6 have body draft of 0.040m with case 5 being the non-stratified equivalent to case 6.

Conclusion

The modified MOFOR has been shown to work well for cases where a body is moving under the action of specified forces. For the current simulations, the significant non-dimensional parameter is the densimetric Froude Number

$$Fr = \frac{U}{(g'L)^{0.5}}$$

Where U and L are velocity and length scales and g' is the reduced gravity value

$$g' = \frac{\Delta\rho}{\rho}\, g$$

ρ is a reference density, $\Delta\rho$ the density difference between upper and lower layers and g the acceleration due to gravity.

These results can be related to an ocean environment with the same layer densities (fresh water, 1000 kgm-3 overlaying sea water, 1027 kgm-3) with the same value of Fr.

For example, this may be the case for a river discharging into the sea, a fjord or loch. Then, for example, if the upper layer depth is assumed to be 20m, a vessel travelling at 4 to 5 knots would be expected to experience some of the effects to those of the model vessel and this would be most problematic for vessels with limited power (e.g. sailing boats).

References

Hornby R P. PHOENICS Modelling of a Moving Body in a Stratified Tank. PHOENICS Newsletter Winter 2019.

Hornby R P. PHOENICS Modelling of the ‘Dead Water‘ Effect. PHOENICS Newsletter Spring 2009.

Dr R P Hornby email: bob.hornby007@gmail.com

CHAM Announcement

Harry Claydon, with input from others at CHAM, has been working on a simulation of the breathing of two people seated opposite each other across a table in a restaurant or pub. There is a background air current of 0.5 m/s from left to right in this view. The two people are 2 m apart from each other. The person on the left is modelled breathing through the mouth with the breathing rate a sinusoidal function with a period of 4 seconds (15 breaths per minute), with a tidal volume of 500 ml. The mouth is modelled as a rectangular slot 3 mm tall by 40 mm wide, giving an area of 1.2e-4 m^2.

The "blue bubble" in the animation shows an isosurface at which the droplet concentration is 1% of the value at the mouth where the breath is exhaled. Where the bubble disappears during the animation, the concentration of droplets has fallen below 1%.

This concentration can be quantified in terms of the number of droplets present by reference e.g. to Papineni and Rosenthal (1997). Figure 5 of that paper shows the number of droplets exhaled in 10 breaths; for mouth breathing, the number of droplets per breath that are ≤1 µm or smaller is 55 droplets.

References

R.S.Papineni and F.S.Rosenthal (1997). The Size Distribution of Droplets in the Exhaled Breath of Healthy Human Subjects. *Journal of Aerosol Medicine. 10 (2)*

J.K.Gupta, C.H.Lin and Q.Chen (2010). Characterizing exhaled airflow from breathing and talking. *Indoor Air (Wiley), 20(1)*, pp 31-39

Engineering CFD Predicts Effects of Ventilation and Aerosol Droplet Size on Associated Virus Spread, by Drs. Vladimir Agranat and Sergei Zhubrin, ACFDA, Thornhill, Ontario, Canada, May 2020.

By extending PHOENICS/FLAIR, a special-purpose engineering software for Heating, Ventilation and Air Conditioning (HVAC) systems , Applied Computational Fluid Dynamics Analysis (ACFDA) has been able to make an important step towards the development of Computational Fluid Dynamics (CFD) computer model for specialized human environments with special emphasis on predicting the virus spread by aerosol clouds emerging from their occupants. Employing this model, a detailed multiphase flow and aerosol particle distributions can be economically predicted by engineers with little or no special training. This will permit the computational assessment of epidemiological safety of built occupied environments prior to construction of HVAC systems, support the design decisions for changing the existing and to-be-constructed projects and reduce the cost and time-scales of testing and appropriate equipment procurement for workplaces and public spaces. It will enable governmental agencies to develop proper health and safety mitigation measures aimed at reducing a possible virus spread after relaxing the lockdown restrictions.

A major concern with current COVID-19 dynamics is that a widely accepted 2-m separation distance could be insufficient. Also, relaxing the lockdown restrictions might cause increases in infections. Moreover, the existing HVAC systems need to be evaluated to be adequate for preventing the virus spread in public spaces.

The purpose of this article is to provide the reader with information about reasonably simple yet realistic and comprehensive method by which he or she may mathematically predict the spread of infected aerosol in a built occupied environment with special emphasis on effects of ventilation rates and sizes of aerosol particles. It is proposed here to apply a CFD technique, in 3D mode, to model a spread of virus by aerosol clouds emerging from human occupants. A detailed CFD modelling of workplaces and public spaces with a specified HVAC system, barriers and people's locations would be helpful in decision making for developing and implementing the proper health and safety mitigation measures.

The aerosol model employed allows to simulate multiphase clouds (air with droplets). Basic details of model theory and its validation are provided elsewhere. The conservation equations for mass fractions of various droplet size groups are made size dependent with an account of liquid-air mixing,

The model has been applied to CFD predictions of aerosol-attached virus spread in a typical workplace ventilated by two exhaust fans shown as grey boxes. It is assumed that the 10-micron and 100-micron droplets are capable of virus spreading from an infected (left) person towards unaffected (right) worker hiding behind a screen. On **Fig. 1-3**, the iso-surfaces of computed 3D variables, C7 and C8, are shown in blue. C7 and C8 are the relative mass fractions (with respect to their values at the source) of those droplets respectively. The displayed 1% and 0.1% clouds bound/contain gas volumes with corresponding C7 and C8 values larger than 0.01 or 0.001.

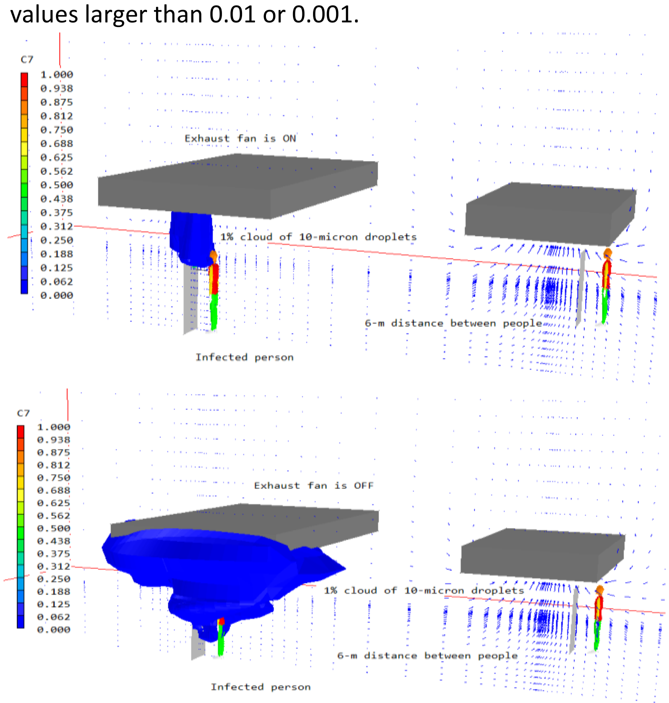
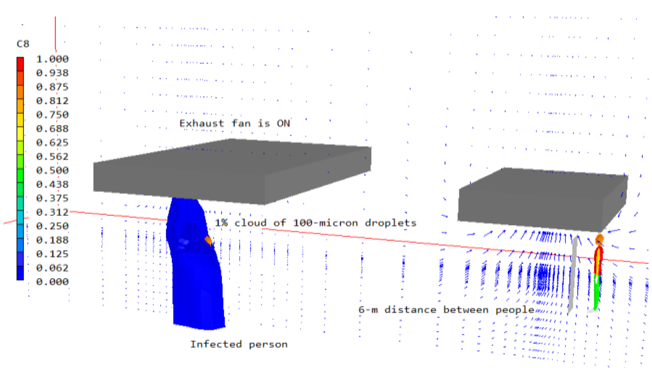


Fig. 1 Spread of cloud of 10-micron droplets with and without ventilation



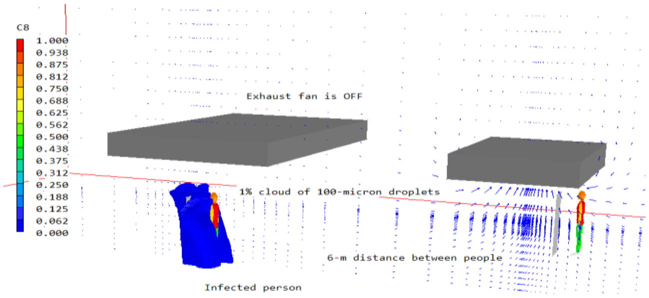


Fig.2 Spread of cloud of 100-micron droplets with and without ventilation

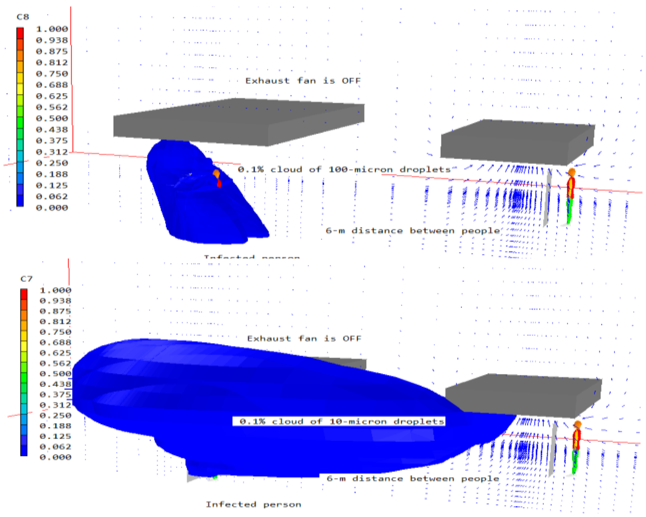


Fig.3 Spread of 0.1% clouds of 100-micron and 10-micron droplets without ventilation

The inspection of the results provided above on Fig. 1-3 appears to the present authors to justify the following immediate conclusions: (a) all the results reported are plausible; (b) they have also been attained easily and economically; (c) the results give an insight into the influences of ventilation and sizes of aerosol particles on spread of virus attached to the latter.

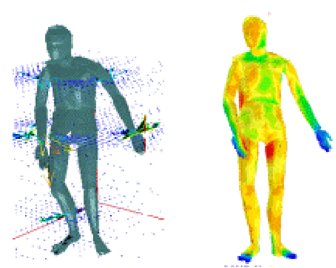
Based on these predictions and successful validation of underlying method reported elsewhere one can begin using engineering CFD in the development of safe designs of ventilation systems. The most important advantage of this approach is that it allows engineers to evaluate new design alternatives in a fraction of the time that was required in the past.

All they have to do is to change the parameters of the model, such as geometry of workplace, ventilation locations, operating conditions, etc., and re-run the analysis in order to determine the effects of the design change they are considering. Using economical CFD simulation, engineers can now evaluate far more alternatives and determine how proposed designs perform.

Both the development cost and development time of safe ventilation system can be substantially reduced as well. Fewer physical prototypes and experiments are needed because engineers can now eliminate ineffective designs using simulation so that only a few optimized designs need to be tested.

The CFD modelling in its PHOENICS/FLAIR embodiment is recommended as a tool for developing the effective health and safety mitigation measures (proper separation distances, ventilation, barriers, masks, etc.). The application of modelling technique can obviously be of great help in identifying the locations of hotspots of largest aerosol agent (and associated virus) concentrations and by this providing an important insight into safe organization of working environments and public spaces.

The other applications of extended CFD methodology related to the current topic could include the evaluation of different scenarios and case studies such as a response of human thermo-physiological system on conditions of Personal Protective Equipment (PPE) exposed to the hazardous environment and a physiological response of protective clothing on stationary and moving persons with effects of physico-chemical properties of protective cover . The predictions such as shown on **Fig. 4** will reveal the key physiological parameters such as deep body temperature, skin temperature, skin blood flow, sweat rate and skin wettedness - all as functions of the quality of room ventilation and air flow conditioning.



The accumulated experience of its practical applications shows all indications that the engineering CFD technique described may be used to account, satisfactory for decision-making purposes, for practically observed properties of the real-life aerosol clouds and associated viruses emerging from the human occupants of built environments. CFD modelling of workplaces and public spaces will help in developing the right health and safety measures aimed at reducing and/or preventing the virus spread in these built environments.

Development of a sustainable system of decentralized mechanical ventilation for façades: Compatibilizing acoustics and air renovation.

Introduction

The product development presented below is the result of a partnership between CHAM and the International Doctoral Programme in Sustainable Built Environment of the University of Minho in Portugal. The simulations using PHOENICS and PHOENICS-Flair were very important to evaluate the thermal and ventilation performance of a prototype that is being developed by PhD student Marco Aurélio de Oliveira. Such prototype consists of a sustainable decentralized mechanical ventilation system for use in façades, capable of combining the normative requirements of acoustics, ventilation and thermal. A geometry was initially created in PHOENICS to assess air velocity and temperature, as shown in Figures 1 and 2

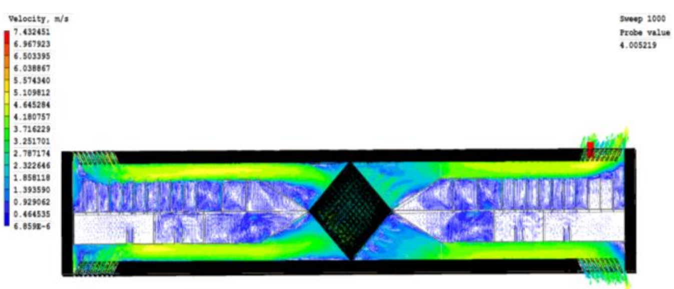


Figure 1 – PHOENICS: CFD air speed simulations

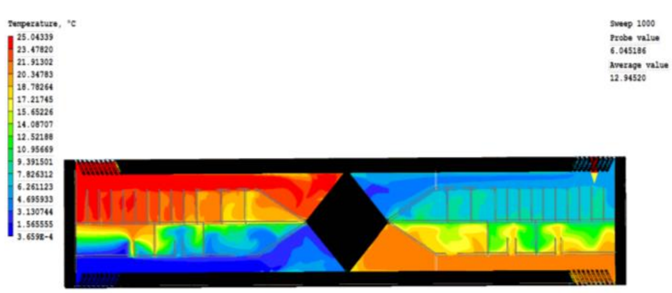
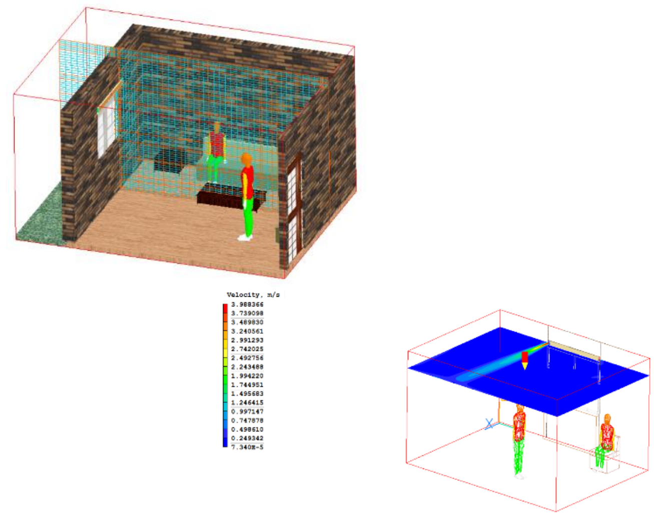


Figure 2 – PHOENICS: CFD temperature simulations

In the central part of the prototype there is a heat recovery system and the driven forces, in the two circuits, arise from two small internal small motors. The results of such simulations were then used in PHOENICS-Flair to evaluate the air circulation velocity as well as thermal comfort conditions in a virtual study environment. Figures 3 and 4 below show some of the results of these CFD simulations.



3 – PHOENICS-Flair: geometrical model and simulation of the air circulation velocity

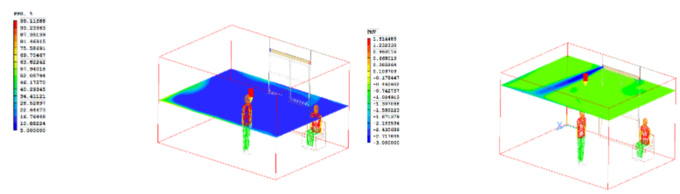


Figure 4 – PHOENICS-Flair: thermal comfort simulation

Results

The results of the simulations showed that the prototype, within the configured parameters tested, met all the ventilation and thermal comfort requirements of the Portuguese standard NP 1037-1 and the standard CEN 15251-2006.

Conclusions

CFD simulations with PHOENICS permitted adjustment and modification of the shape and materials of the prototype until it resulted in the final design solution to become the guideline for the manufacture and subsequent evaluation of a physical prototype in the laboratory. Thus, it was possible to establish an effective and efficient design solution during product development whilst avoiding the costs associated with the construction and physical testing of preliminary designs. The use of PHOENICS and PHOENICS-Flair was central to our work as researchers and was a powerful and advanced tool in this regard. We would like to thank the CHAM Company and especially Mr Peter Spalding, without which this stage of our research would not have been possible.