PHOENICS News

CHAM

"We fully endorse PHOENICS as one of the best CFD codes in the world. For many years we have been able to solve important engineering and scientific problems using the code. Many BSc, MSc, PhD theses have been finalized using PHOENICS, from which many scientific papers have been published in the last years. Furthermore, we are also very satisfied with CHAM's Technical Support services."

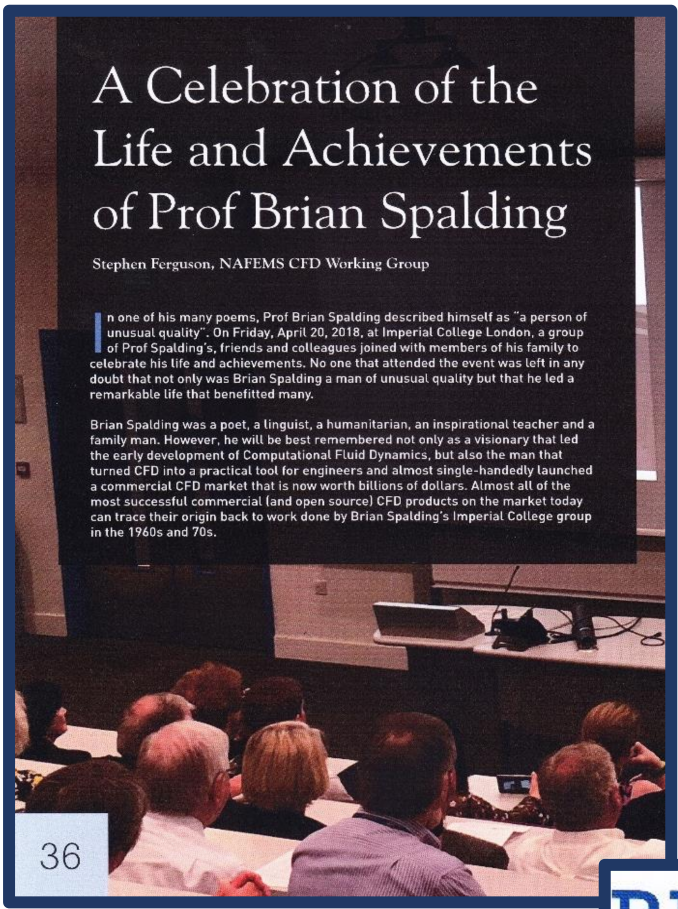
- *Gazi University, Dr. Senol Baskaya*

This Edition includes:

Background Photograph: © Robynne Leigh Moynihan

Summer 2018

A Celebration of the Life and Achievements of Professor Brian Spalding, by Stephen Ferguson, NAFEMS Working Group.



A Celebration of the Life and Achievements of Prof Brian Spalding

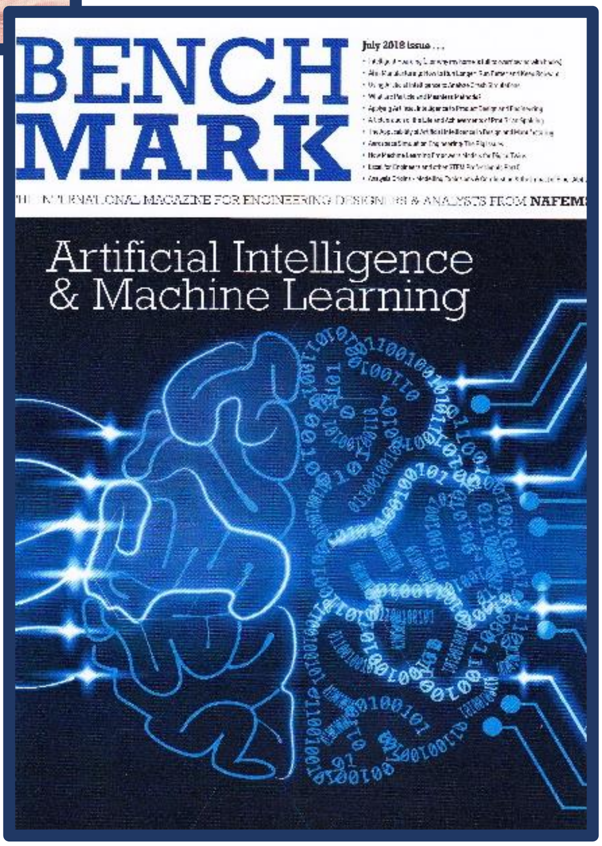
Stephen Ferguson, NAFEMS CFD Working Group

In one of his many poems, Prof Brian Spalding described himself as "a person of unusual quality". On Friday, April 20, 2018, at Imperial College London, a group of Prof Spalding's, friends and colleagues joined with members of his family to celebrate his life and achievements. No one that attended the event was left in any doubt that not only was Brian Spalding a man of unusual quality but that he led a remarkable life that benefitted many.

Brian Spalding was a poet, a linguist, a humanitarian, an inspirational teacher and a family man. However, he will be best remembered not only as a visionary that led the early development of Computational Fluid Dynamics, but also the man that turned CFD into a practical tool for engineers and almost single-handedly launched a commercial CFD market that is now worth billions of dollars. Almost all of the most successful commercial (and open source) CFD products on the market today can trace their origin back to work done by Brian Spalding's Imperial College group in the 1960s and 70s.

36

On April 20 there was a Celebration of the Life of Brian Spalding wonderfully arranged by Professor W P Jones, of Imperial College, and held there. It was a day when over 100 people gathered to remember someone who had impacted on their lives in different ways. Some knew him as a colleague, some were his students, others knew him as man rather than scientist. Professor James Sterling, Provost of IC, had not met Brian but his welcome clearly indicated that he had taken time to learn of him prior to the event. Andrew Pollard led the remembrances *(Brian Spalding – The Early Years*) followed by Brian Launder (*Recollections of a Junior Lecturer*), Marcel Escudier *(Memories of DBS and Others*), Jim McGuirk (*Personal Recollections of DBS as Teacher and PhD Supervisor*), Derek Bradley – an amazing nonagenarian *(Brian Spalding and Combustion Lift-Off*), Alan Swanson (Brian Spalding as a Trade Union Leader – this was news to me!), Norberto Fueyo (*Few Painters Invent New Colours – Brian Spalding and Turbulent Combustion Modelling*) then we broke for lunch and Brian's family had time to meet a small proportion of those who had attended. After lunch others remembered: Steven Beale *(Brian Spalding - A Latter-day Student's view*), Sergei Sapozhnikov by video from St Petersburg (*The Poetry of Brian Spalding*), Said Elghobasi (*The Telegram that Changed my Life*), David Gosman (*Brian Spalding & the Origins of Commercial CFD*), Suhas Patankar *(Brian Spalding's Profound Impact on CFD*), John Ludwig & Mike Malin (*CHAM*), Wolfgang Rodi (*Brian Spalding & ERCOFTAC*), Kemal Hanjalic (*Brian Spalding & the ICHMT*), Jana Levich (*The Levich Affair*) and myself (*Life with Brian*). In the audience was a gentleman who did not know Brian but had travelled from Scotland to attend. Afterwards he wrote an article for Benchmank the NAFEMS Journal.



The day recalled the diversity of Brian's life and the richness of his mind. This outline is a welcome reminder. Colleen Spalding

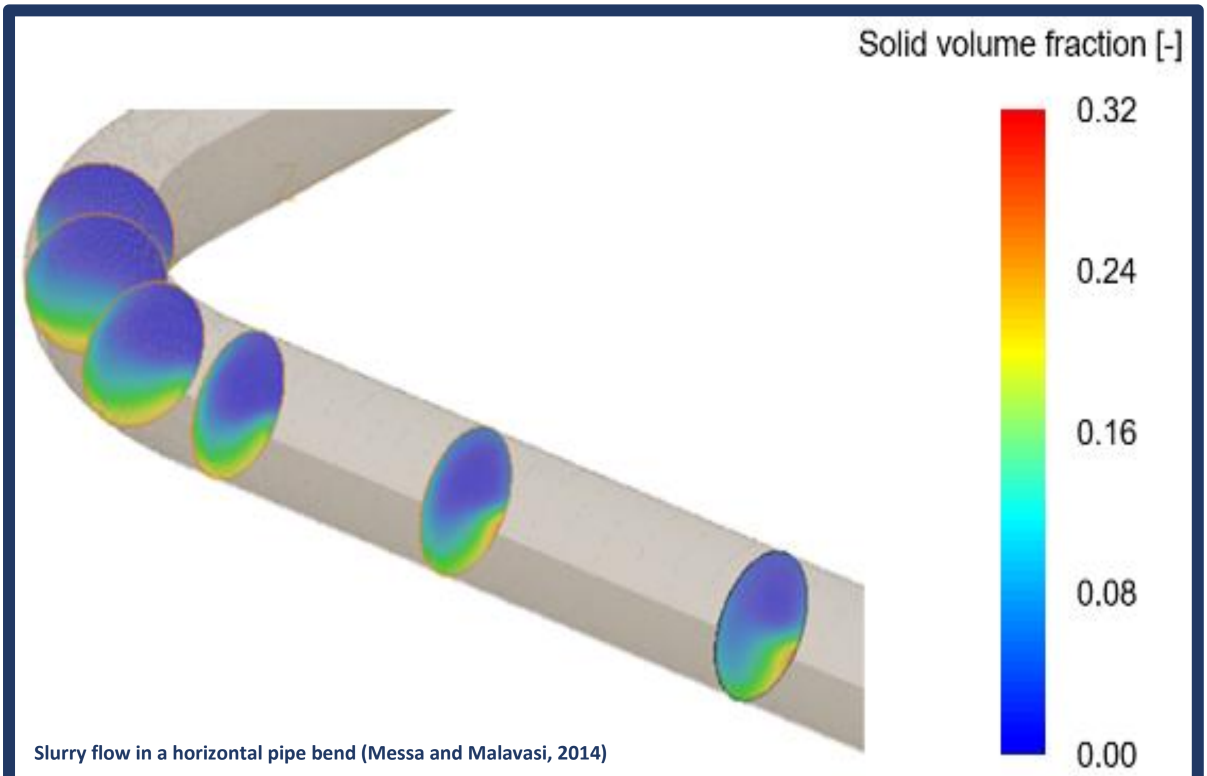
PHOENICS aids continuing research on Hydrotransport Processes

Gianandrea Messa, Stefano Malavasi, FluidLab Research Group, Politecnico di Milano

The following article describes the CFD simulation of water/slurry flows in horizontal pipelines; outlining recent achievements and new perspectives by researchers at the FluidLab research group of Milan Polytechnic.

In recent years, the FluidLab research group at Milan Polytechnic has developed expertise in the modelling of hydrotransport processes of interest in the mining industry. Research activities on this topic date back to work undertaken by Gianandrea Messa in 2010 on development of a CFD model for the simulation of turbulent, liquid-solid, slurry flows in horizontal pipes. The model was implemented in PHOENICS by extending the simulation capabilities of the built-in, two-fluid IPSA (Inter-Phase Slip Algorithm) model in the code.

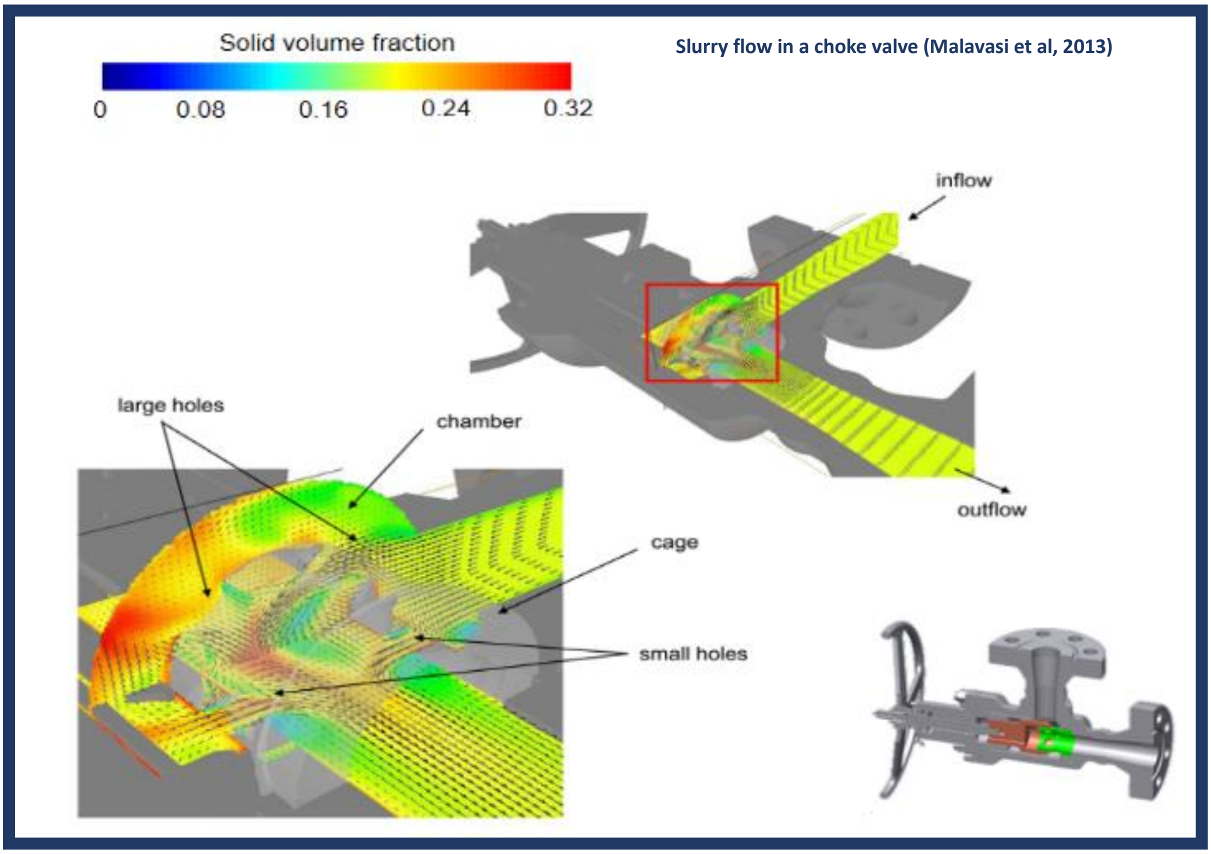
Validation against experimental data reported in the scientific literature proved the model's capability to predict accurately the features of slurry-pipe flows of great relevance to engineering, such as pressure losses and concentration distribution, under conditions where turbulence maintains the solids in suspension and no particle accumulation occurs. Strengths of the developed model, compared to similar ones, reside in its simplicity, the few input parameters involved, its accuracy over a wide range of flow conditions, and its high numerical performance facilitated by the efficient algorithms embedded in the PHOENICS code.



Slurry flow in a horizontal pipe bend (Messa and Malavasi, 2014)

The two-fluid model acquired international recognition in the scientific community, as evidenced by numerous publications in scientific journals and in the proceedings of international conferences. Furthermore, in 2013, Gianandrea Messa won the GII award for the best PhD thesis in Water Engineering from the Italian Group of Hydraulic Engineers; in the same year, he was among the three Italian finalists of the "Da Vinci Competition" organized by ERCOFTAC. In subsequent years, the modelling of slurry flows has always been a strong topic of research within the FluidLab group, advantaged by a fruitful cooperation with CHAM. The two-fluid model was further improved, and related phenomena of significant engineering interest were explored, such as impact erosion produced by particle-wall collisions. As part of a research contract financed by ENI S.p.A., an Italian oil and gas company, dedicated laboratory facilities were set up in the Hydraulic Laboratory of Milan Polytecnic, and were used to validate the numerical results. In 2017, Dr Messa received the “Hydrotransport Next Generation Award” in recognition of his “significant scientific / engineering accomplishments and his contribution to the principles / applications underlying hydrotransport technology before the age of 40 years.” He described his main research achievements in an invited keynote lecture at the 20th International Conference on Hydrotransport in Melbourne, Australia.

Joining the Hydrotransport community gave the FluidLab research group the opportunity to explore the boundaries of knowledge in the field, giving a new impetus to the group’s research on particle-laden flows. In particular, scientific cooperation has been established recently with Professor Vaclav Matoušek, who is affiliated to the Czech Technical University in Prague and the Institute of Hydrodynamics of the Czech Academy of Sciences who is considered one of the most prominent slurry-flow experimentalists. The synergy between the numerical and experimental approaches, engendered via the expertise of the two parties, will provide greater insight into the physical phenomena underlying slurry transport, thereby improving modelling capabilities and making CFD an effective engineering tool for this type of application.



Slurry flow in a choke valve (Malavasi et al, 2013)

Recent results-focused on the characterization of the velocity field of liquid-solid mixtures flowing in horizontal pipes-will be presented at the International Conference on Conveying and Handling of Particulate Solids (CHoPS), which will be held at Greenwich University, Maritime Campus, London, September 2018. http://www.constableandsmith.com/events/chops-2018

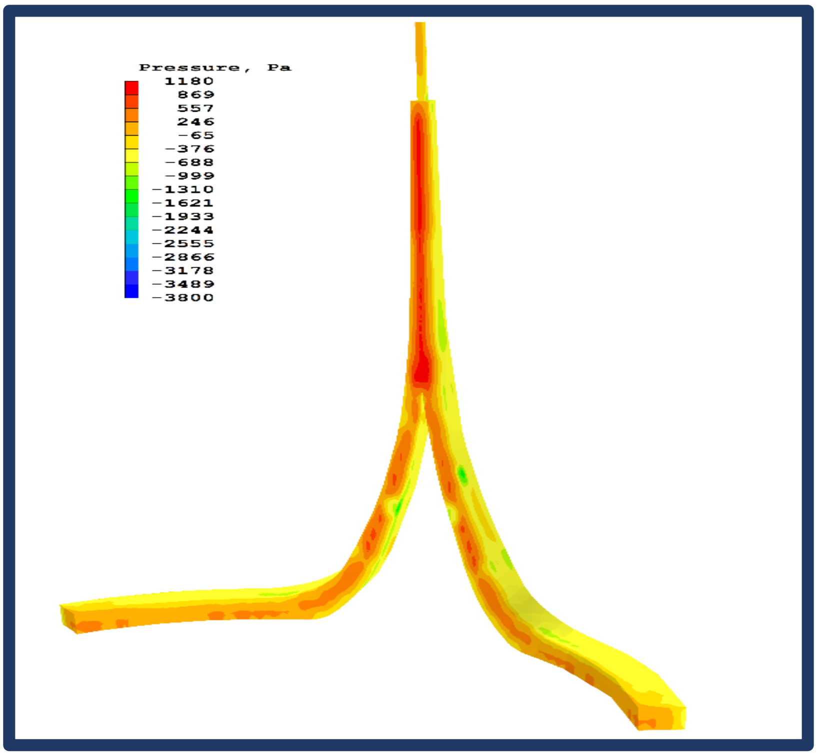
For more information, visit: www.fluidlab.polimi.it www.cham.co.uk; Email: marketing@cham.co.uk

Related article: Concentration Distribution and Pressure Gradient of Particle-Water Slurry Flows in Horizontal Pipes www.cham.co.uk/DOCS/newsletter_archive/phoenicsnewswinter2011.pdf - Page 4

Wind Flow Through the Çanakkale Antenna Tower, Turkey.

by Kadir Ozdemir, Simutek.

Pressure Loads on the Tower



Çanakkale Antenna Tower is a 100-meter Observation and Broadcast Tower in Çanakkale, western Turkey.

The most sensitive part of the project is the tower which is 95 meters high. Because of the height and slenderness of such towers resonance, and excessive vibration problems, can occur under wind loads.

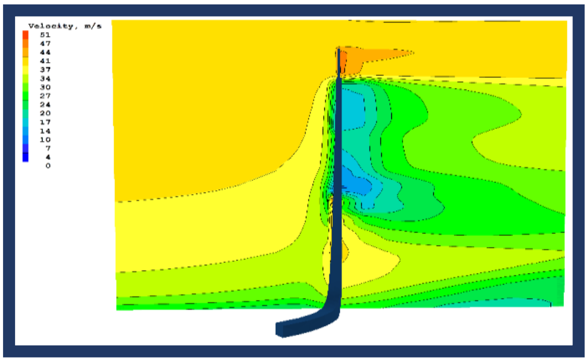
In the lower atmosphere, wind speed generally increases along a gradient as height increases. However, the primary concern related to the forces that wind exerts on a tower is not speed but pressure. Wind loads have significant pressure effects on high buildings. The effects of wind pressures on a structure are a function of the characteristics of the approaching wind, geometry of the structure under consideration, and geometry and proximity of structures upwind.

The model was created by importing the tower geometry as an object from CAD. The 'Wind' object was added to specify wind characteristics in terms of direction, strength and profile.

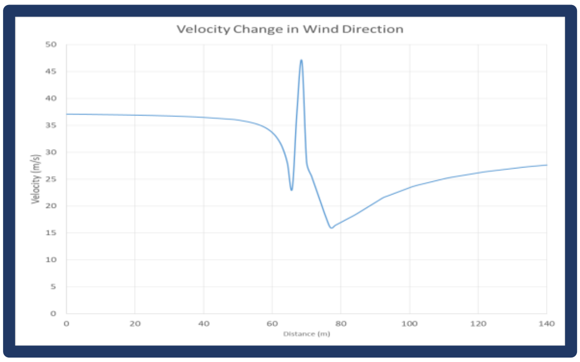
Results were obtained in 24 hours with a 12-processor parallel solver. Graphs were created with values from "plot variable profile" options.

For demonstration purposes, the following assumptions were made

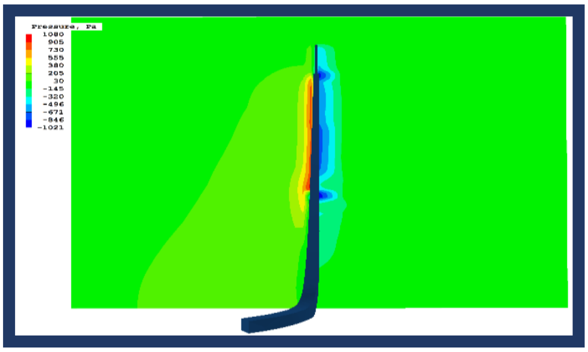
- The simulation was set up using steady-state incompressible flow
- KE-Standard turbulence model was used
- Wind speed is 40 m/s at reference height 95 m



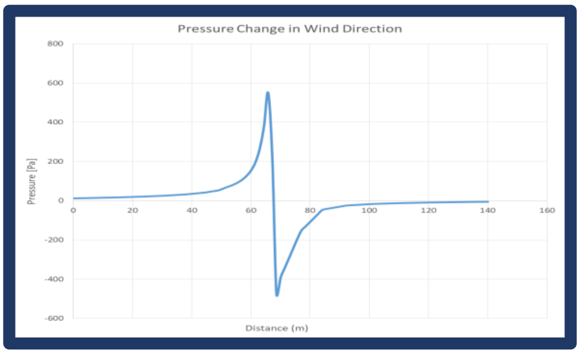
Velocity Contours in Wind Direction



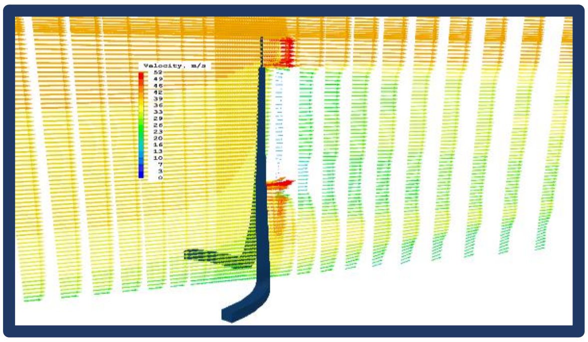
Velocity Change in Wind Direction at 40 m height



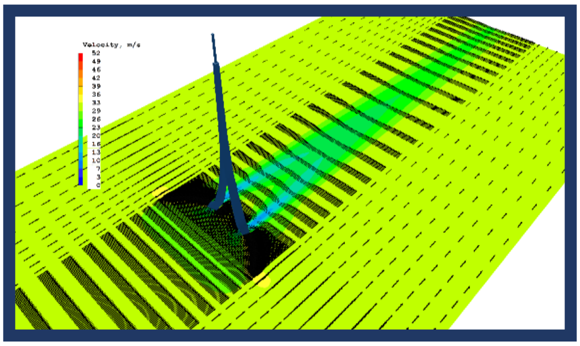
Pressure Contours in Wind Direction



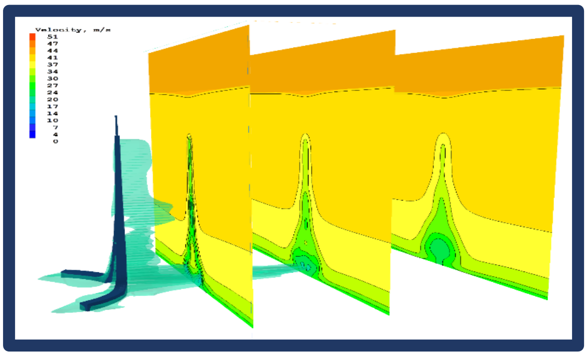
Pressure Change in Wind Direction at 40 m height



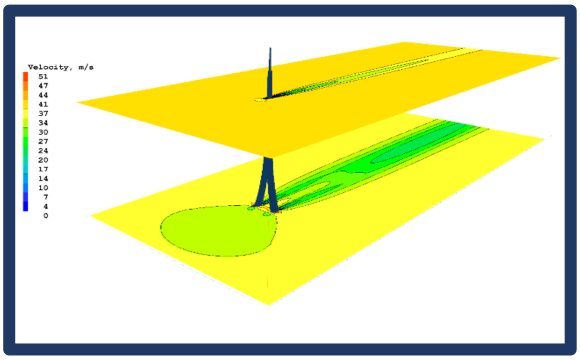
Velocity Vectors in Wind Direction



Velocity Vectors in Wind Direction



Velocity Contours with Time in Wind Direction



Velocity Contours Change with Height

Trombe Wall Case Study

By Andrew Carmichael and Ryan Dyer – CHAM

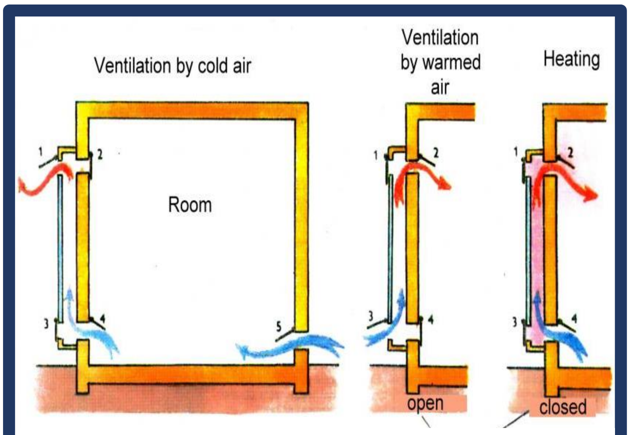


Figure 1 Visual description of the physical principles behind using a trombe wall as a natural ventilation method.

Figure 2 The Trombe wall ventilation system (left) has been modified to be used both for heating and cooling using renewable energy sources. Researched by Marwa Dabaieh from Lund University in Sweden. Tested in Saint Catherine, Egypt Photo: Insaf Ben Othman

A Trombe wall consists of a sun-facing, solid concrete or masonry wall behind a glazed space. When the sun shines, energy comes through the glass and is stored in the wall's thermal mass. When the sun sets, or is blocked, and the temperature drops the wall releases heat into the room behind. Trombe walls are commonly used to absorb heat during winter sunlight hours and slowly release that heat at night, when it is most needed.

Solar heat's higher-energy ultraviolet radiation has a short wavelength which passes through glass almost unhindered. When this radiation strikes a wall or slab, the energy is absorbed and then re-emitted in the form of longer-wavelength infrared radiation. Infrared radiation does not pass through glass as easily, so heat is trapped and builds up in the enclosed space.

A secondary benefit of this system is the ability to use it to drive a passive convection current though the room to which it is attached. As seen in Figure 1, the heated wall creates a buoyant current in the glazed space which draws cool air from the floor and heats it as it rises. This provides a second convective heating method to the main radiative one.

Interestingly, this convection current can also be used to provide ventilation and cooling if warm air is not reintroduced into the room. In this case, the current can draw in air from a cooler, shaded, side of the house. CHAM's RhinoCFD was used to demonstrate this principle using a small scale example, where a simple room was set up with a Trombe wall on one face and a window opposite. The Trombe wall was

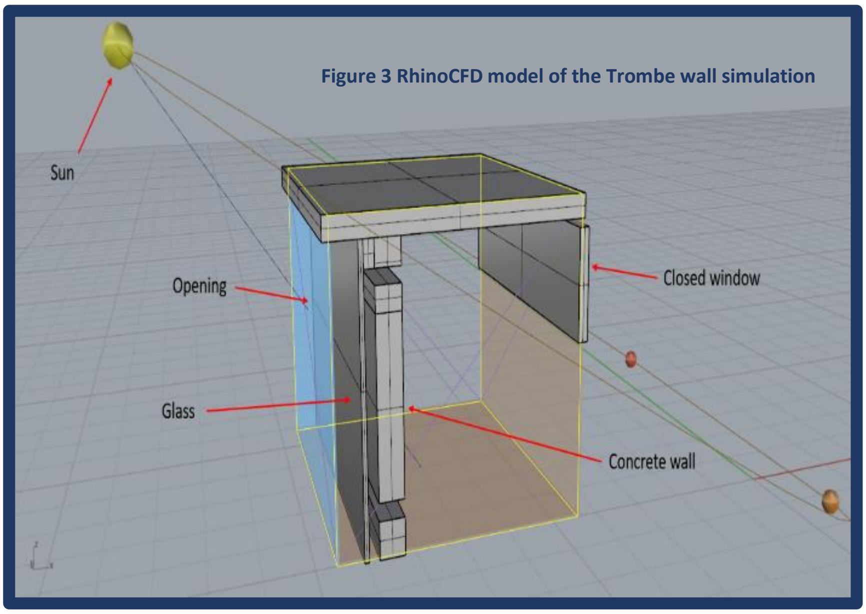


Figure 3 RhinoCFD model of the Trombe wall simulation

designated concrete and the glass panel transparent to the incoming sun (set up with the inbuilt sun model).

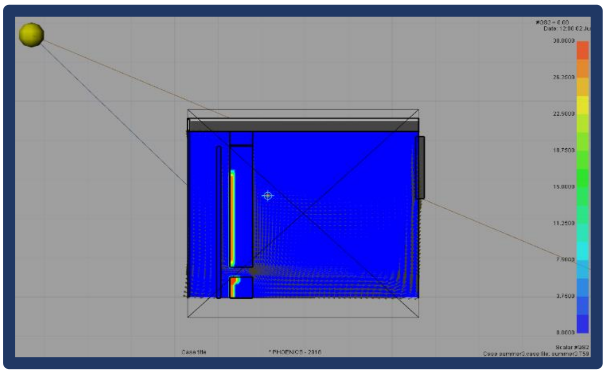


Figure 4 RhinoCFD's inbuilt solar model easily calculates and displays the areas upon which the solar radiation impinges based on the latitude and time of day that are prescribed

The simulation was run in transient mode for 100 minutes, with a step size of 2 minutes, which was small enough to ensure convergence. Given the symmetrical nature of the geometry, the case was run in 2D, with only one cell in the Y-direction. This case ran for winter "room warming" and summer "room cooling" and its effectiveness was demonstrated.

For the winter case, the ambient temperature was set to 15 degrees at the start of the simulation, with direct solar radiation set to 800W/m^2. This flux was applied only to the areas directly illuminated by the sun; it was assumed there was no diffuse radiation present. RhinoCFD's solar model automatically calculates where this area should be, based on prescribed solar elevation and time of day. This area changes with the movement of the sun as time advances, but in this case the variation is minimal.

Simulation Setup Details

The mesh used comprises 2240 cells, with up to 5 cells in the glazed space to capture accurately the convective plume in the area. The KE-Chen turbulence model and Immersol radiation model were selected with wall emissivity set to 0.9.

In the summer case, aside from the change in geometry to allow the warm air to escape the glazed space, and the ambient temperature being set to 25C^o, all other conditions were as per the winter case.

Results

Winter Case

Looking at temperature distribution inside the room after 100 minutes, it can be seen that it increased substantially from 15C^o to about 20C^o, with a visible current through the glazed space.

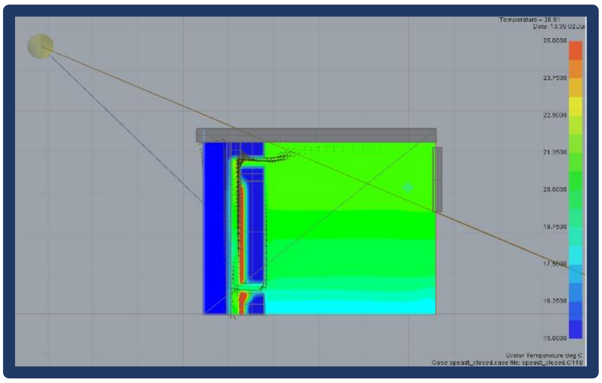


Figure 5 Temperature distribution in the room for the winter simulation after 100 minutes.

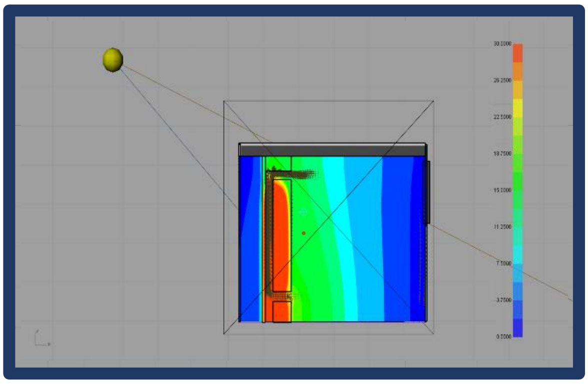


Figure 6. Radiant temperature in the room from a steady-state simulation.

Another interesting aspect is the effect of radiation in the model. A separate steady-state simulation looking at this shows how the radiative temperature calculated by the model behaves as expected: the trombe wall accumulates energy and heat is trapped by the glass façade in front of the wall. Heat is then able slowly to penetrate into the room, helping to warm it.

Summer Case

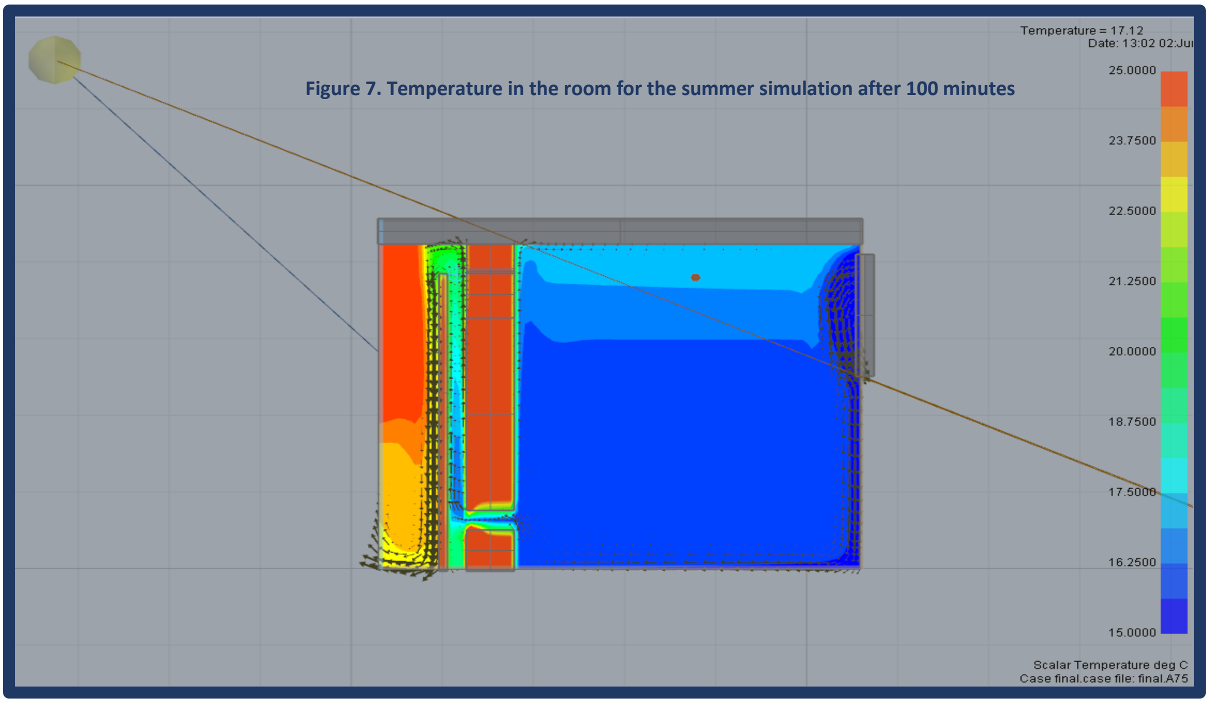


Figure 7. Temperature in the room for the summer simulation after 100 minutes

In the summer case, it can be seen that, despite setting the initial temperature in the room to 25C°, given a source of cool air (imagined to be from a shaded area of the house or an area with some active cooling), the Trombe wall has the effect of creating a current in the room that will help draw in cool air and displace warmer air.

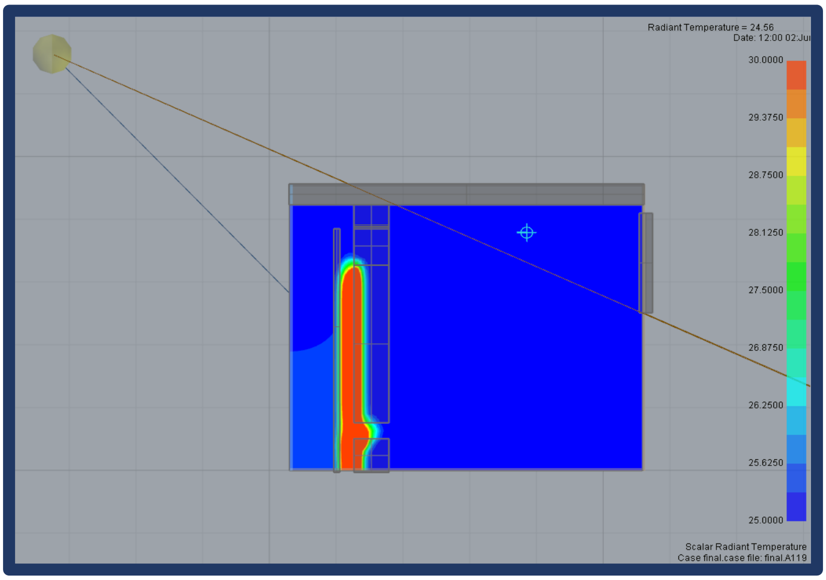
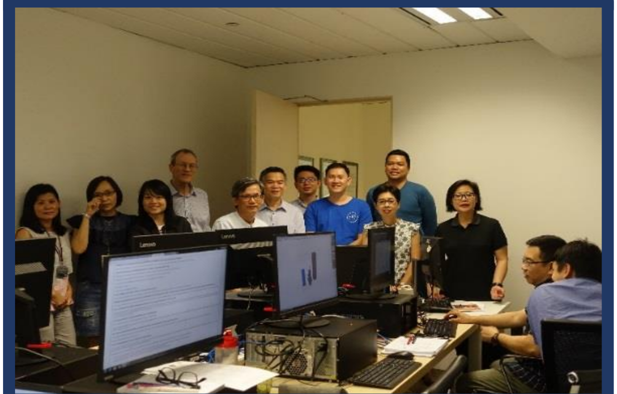


Figure 7. Radiant temperature in the room after 100 minutes in the summer simulation.

The effect of radiant temperature can be viewed, at the end of the 100 minutes run time, to ascertain its effect. Figure 8 shows that most of the radiant temperature is contained in the glazed portion of the Trombe wall and not much has penetrated the room. This corroborates the initial phase-delay supposition, whereby the trombe wall does not radiate much heat during the day while it is being illuminated (due to heat still being absorbed by the wall), but instead releases it mostly at night, helping to maintain ventilation and even out temperatures in the room.

http://www.cham.co.uk/_docs/casestudies/Trombe_Wall_Case_Study.pdf

News from CHAM: Dr David Glynn visited Hong Kong and Singapore on behalf of CHAM to deliver training courses and visit clients.



Hong Kong, April 16 – 20

The Training Course in Hong Kong was attended by a group of persons from five institutions. Some were experienced PHOENICS users updating their knowledge, some were new users and some were in-between. The group comprised engineers and architects whose fields of interests included wind problems and transport of pollutants due to wind - in particular pollution arising from traffic in urban street canyons. All who attended seemed to enjoy the experience. One long-term PHOENICS User said that it renewed her passion and enthusiasm for CFD.

At the course, and during ensuing visits, new features of PHOENICS-2018 V1 were presented. Potential users were particularly interested in the new Pedestrian Wind Comfort feature incorporated in PHOENICS 2018, and in Heat Island Modelling. There was also interest in problems associated with siting sewage processing plants in close proximity to residential blocks which could cause odour problems depending on wind direction.

Singapore, May 14 - 18

The training course, held at a Company of Architects and Engineers was attended by twelve persons who participated in the lectures and associated workshops. A major interest was in modelling ventilation and wind particularly with regard to modelling flow through louvres because large louvre walls admit natural ventilation. There was also an interest relating to data centres.

PHOENICS Users visited after the course were interested in modelling louvres, wind flows, condensation, and wind-driven rain. Interest was expressed in the Urban Heat Island and heat dispersion in a transformer room.

News from CHAM – A visit to Shanghai and Tokyo.
By Colleen Spalding

In April 2018 Colleen Spalding visited Shanghai – Feiyi, CHAM's Agents in Shanghai and CHAM Japan, CHAM's branch in Tokyo. She was accompanied by Dr Jeremy Wu (now based in Shanghai after many years at CHAM in London).

It was most enjoyable to meet colleagues from Shanghai-Feiyi and to participate in interesting discussions of a general, technical and business nature. In addition to formal conversations there was time for a river trip to admire the Shanghai skyline and a visit to a Village which showed how life in the area had been before the City encompassed many such small conurbations as it grew to its current size of 30 million inhabitants. Many thanks to Mr Fan, and all of his staff, for their hospitality and for making us feel so welcome. I look forward to returning in the not-too-distant future.

After two days in Shanghai it was time to renew acquaintanceship with colleagues at CHAM Japan in their new offices. Discussions were held on technical and sales activities, and about future plans. A most interesting visit was made to Shibaura Institute of Technology where a fascinating PHOENICS demonstration was given, and conversations held with a Professor at the University who has been a PHOENICS User since 1987 and a senior researcher from Mitsubishi Materials who uses the software to model extrusion processes. The informed and enthusiastic attitude of CHAM-J staff made our stay a pleasure. It was particularly satisfying, on the last evening, to join staff in the office after work which gave the opportunity for more relaxed conversations of a less technical nature.

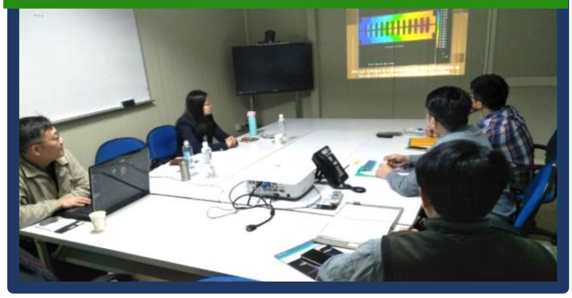
Thanks are due to Mr Kong and all his staff for making us feel welcome not only during business and technical discussions but during a trip to Kyoto on the Shinkansen and evenings spent exploring the City and remembering how delicious Japanese food is.

CHAM Japan is having a PHOENICS User Meeting on October 19 which will be attended by Dr John Ludwig on behalf of CHAM; I look forward to returning soon.

News from CHAM continued

We would like to take this opportunity to welcome Tom James, as he joins our London office as our latest CAD/Software Development Engineer.

News from CHAM Agents – Champion, Taiwan By Jasmine Li

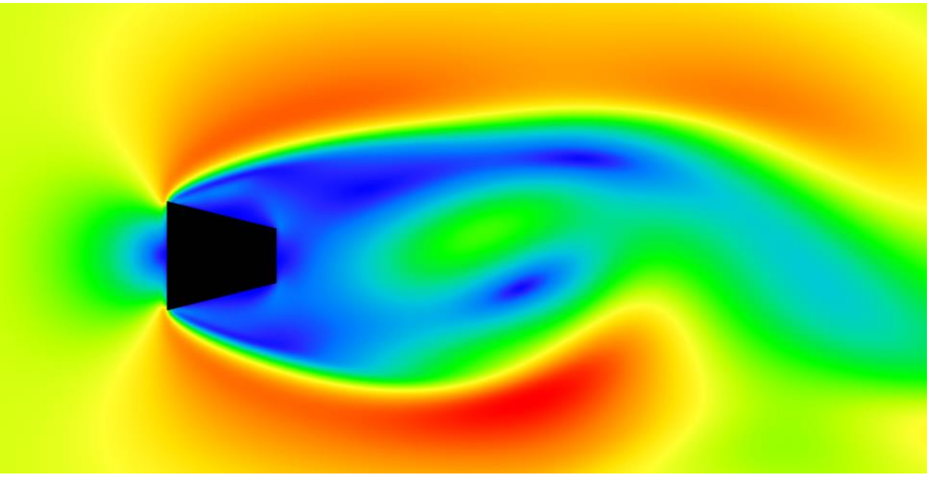


Photograph of a PHOENICS-CVD presentation held at a Technology Energy Company.

PHOENICS CFD Analysis of a Vortex Flow Meter: A transient Analysis using Large Eddy Simulation (LES).

CHAM was contacted by an air-technology manufacturer to help create a PHOENICS CFD model of a vortex flow meter, which could be used to aid design studies of this flow-measurement device.

The PHOENICS Cartesian cut-cell solver was used to simulate air flow through the flow meter using large eddy simulation. Below is a clip from an animation which shows, in terms of the velocity magnitude, the vortices shedding from the trapezoidal shedder housed in the aligning conduit. The frequency of these vortices is used to measure the flow rate on the basis that the Strouhal number remains constant with Reynolds number over the operating range of the device. This means that the shedding frequency is directly proportional to the average flow velocity and flow rate through the conduit.



A key advantage of CFD for this application is that it provides a relatively inexpensive and rapid means of reducing design and development costs. For example, CFD can help in the investigation of different shedder geometries, in ascertaining the influence of upstream disturbances on different designs, and also in performing flow-meter calibration studies.

In this particular case, the main advantage was that CFD enabled a better understanding of the detailed vortex-generation mechanism, which isn't easy in the physical model because of the measurement difficulties associated with the very small bluff body and the transient nature of the vortices.

With CFD the flow field could be viewed in slow motion to find, for example, a good position for the microphone sensors, which are used to detect the pressure pulses from the vortices.

The full animation can be viewed on https://www.youtube.com/watch?v=p1TAYrw6IDk

Contact us:

Should you require any further information on any of our offered products or services, please give us a call on +44 (20) 8947 7651. Alternatively, you can email us on sales@CHAM.co.uk

Our website can be viewed at www.CHAM.co.uk and we are on the following social media:

Concentration Heat and Momentum Limited

Bakery House
40 High street
Wimbledon Village
London SW19 5AU, England.