PHOENICS News

Contents

"We have been using PHOENICS for the last decade for quite a few researches including data center and hospital operating room modeling etc. The tool is most favored by our students and researchers with its strong capability, straightforward interface, as well as superior technical support. "

- *John Zhai, Boulder University, Colorado, USA*

Autumn 2018

Concentration Heat and Momentum Limited, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, UK
Telephone: 020 8947 7651 Fax: 020 8879 3497
E-mail: phoenics@cham.co.uk, Web site: http://www.cham.co.uk

Modelling a Gravity Dust Catcher in a Steelmaking Plant

By Tom James and Michael Malin of CHAM

During steelmaking, blast furnaces release a dust-laden gas flow which enters a dust collection system for gas cleaning and recycling of particulate matter. The dust contains fine particles that are formed from the reactions taking place in the furnace. The main component of the collection system is a either a gravity or a cyclone dust catcher which separates the mixture of dusts from the spent gas flow. Recently, CHAM helped a major steelmaking company build a PHOENICS CFD model of a gravity dust catcher to calculate the particle separation efficiency of an existing design with a view to investigating geometrical modifications to improve efficiency.

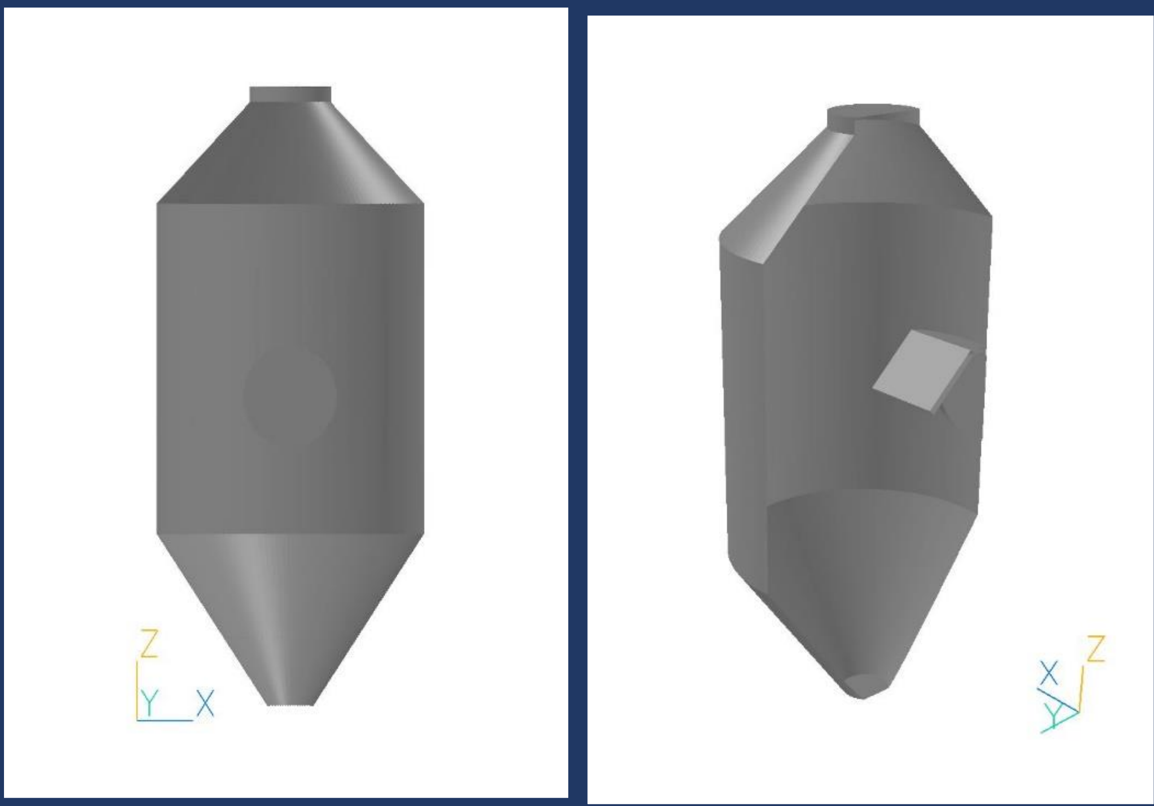


Figure 1: Gravity Dust Catcher –the geometry, including a cut-away perspective showing the internal guide positioned over the inlet.

As shown in Figure. 1, the gravity dust catcher comprises a side-entry inlet section, a main cylindrical separation chamber incorporating a dust-deflector plate, a lower funnel-shaped dust hopper, and an upper conical-shaped transition section leading to the gas outlet pipe. The dust catcher relies solely on gravity to separate dust particles. Flow passes beneath the deflector plate into the main body of the unit before eventually turning upwards in the bottom third of the dust-catcher causing the heavier particles to deposit themselves in the bottom hopper.

The PHOENICS Cartesian cut-cell solver was used to simulate gas flow through the dust catcher and the two-equation Chen-Kim k-ε model was used to model turbulence. The volume fraction of the particulate material is typically less than 2% by mass, so an Eulerian-Lagrangian approach (GENTRA) was used to track this sample of dispersed particles through the flow field. Turbulent fluctuations of the gas phase can be expected to have a significant effect on the trajectory of dust particles; this was accounted for by using a stochastic eddy-interaction model. Separation efficiency was obtained by releasing an even spread of mono-dispersed particles of 10 µm diameter across the inlet boundary, and then monitoring the number escaping through the top outlet boundary.

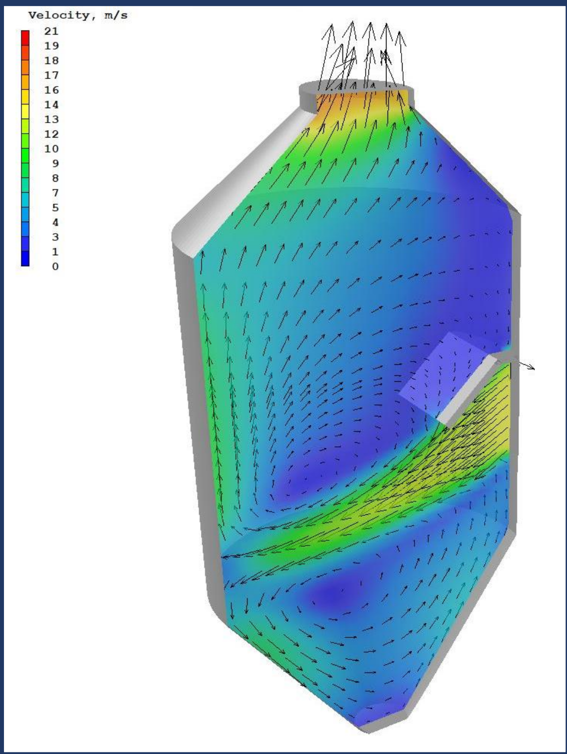


Figure 2: Gravity Dust Catcher – absolute gas velocity contours

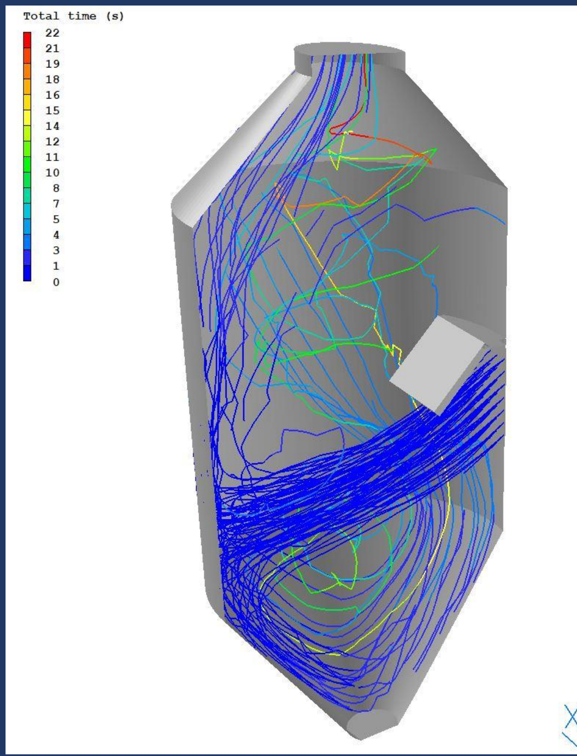


Figure 3: Gravity Dust Catcher – sample of dust particle trajectories coloured by total time

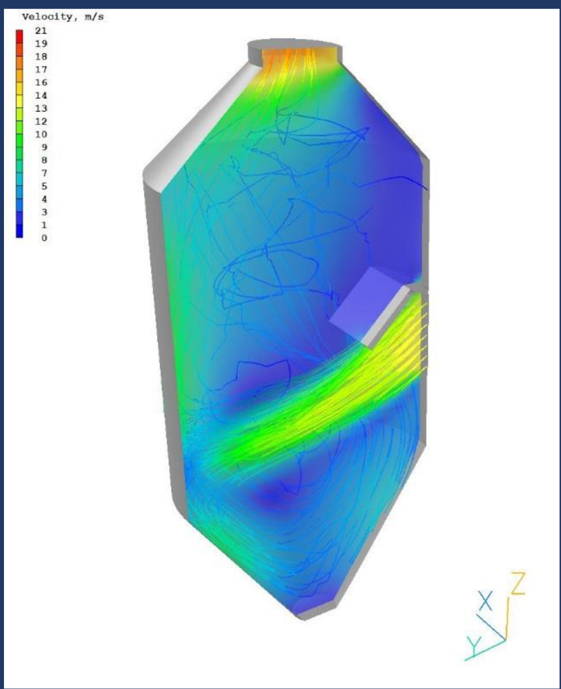


Figure 4: Gravity Dust Catcher – sample of dust particle trajectories coloured by absolute gas velocity

The predicted gas flow in shown in Figure 2 where one can see that the inflow is guided by the deflector plate into the bottom third of the dust catcher, where it then slips into an up-flowing and down-flowing stream after impingement on the far wall.

A sample of the particle trajectories emanating from the inlet is plotted in Figure 3; for this basic design, the CFD model predicts a collection efficiency of 45%.

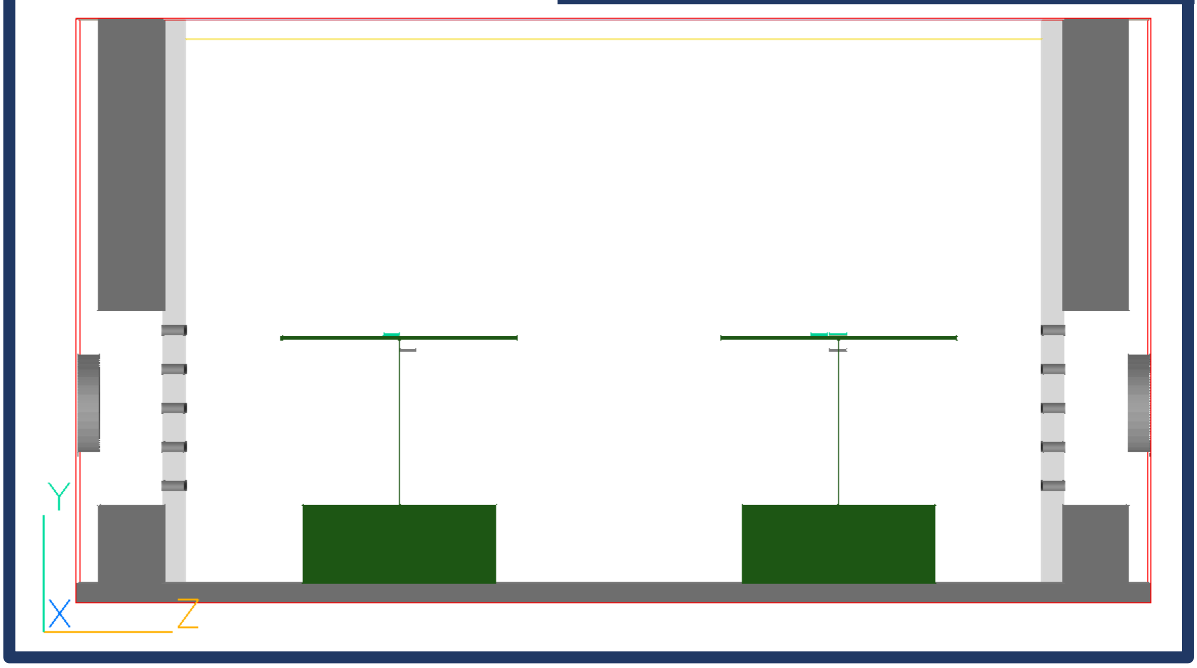
In Figure 3 the trajectories are coloured with time, whereas in Figure 4 they are coloured with the absolute gas velocity.

The next phase of the work is to assess the sensitivity of the model predictions to grid size, particle size distribution, increases in the particle population number and spread across the inlet boundary. Once the simulation of the basic design has been verified for numerical accuracy, the model will be used to explore design alternatives aimed at improving collection efficiency.

Developing Physical Models to Understand the Growth of Plants in Reduced Gravity Environments for Applications in Life-Support Systems.
By Lucie Poulet, University of Clermont Ferrand, France

Miss Lucie Poulet defended her thesis on July 11 2018 at the University of Clermont Ferrand, France (Thesis directors: Prof JP Fontaine & Prof CG Dussap). Her thesis title was: "Developing Physical Models to Understand the Growth of Plants in Reduced Gravity Environments for Applications in Life-Support Systems". PHOENICS 2017, CFD software from CHAM, was used to perform a study of computational fluid dynamics inside an enclosure to ascertain local velocity values at the leaf surface which could not be determined using only the two anemometers.

Figure 4.10: 3D model of the enclosure - Front View.



A 3D model of the enclosure was developed using the actual dimensions of the one that flew in the aircraft (Figure 4.10). The length direction is Z; the width is X; and the height is Y. Walls and unused empty spaces of the enclosure are modelled using blockage objects. Porous plates are modelled using the exact number of holes (19 x 5 per plate) and the outlet is modelled using a porous plate which makes the file less heavy (Figure 4.11). Pots are modelled with cubic blockages of dimensions 9 cm (X) x 4 cm (Y) x 9 cm (Z) to represent the 9-cm diameter and 4-cm height cylindrical pots. The leaves studied are modelled together with 3D printed frames, using plates of the dimensions of the 3D printed frames (11 cm x 11 cm).

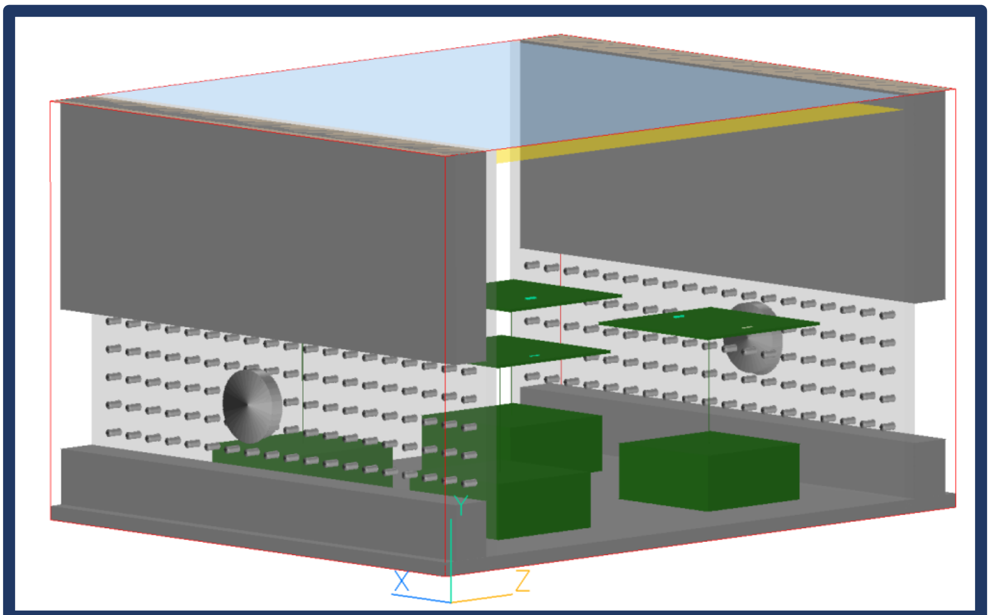


Figure 4.11: 3D model of the enclosure - Side View.

Blockages of 4 mm length and width and 8.5 cm height are used to model the plant stems. Point history objects are used to follow the evolution of air velocity at specific points in the enclosure, one placed for each studied leaf and one for each anemometer, as indicated below.

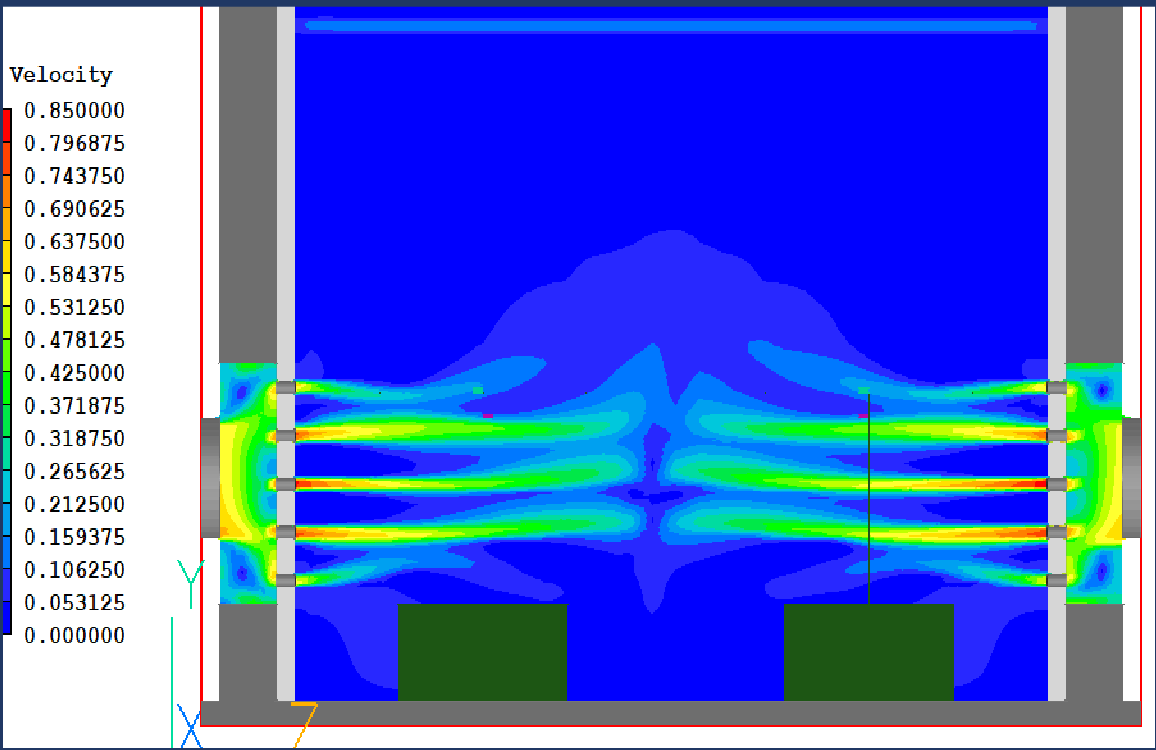


Figure 4.12a: Velocity profile into the enclosure at the lower air flow rate (in m/s) – Front

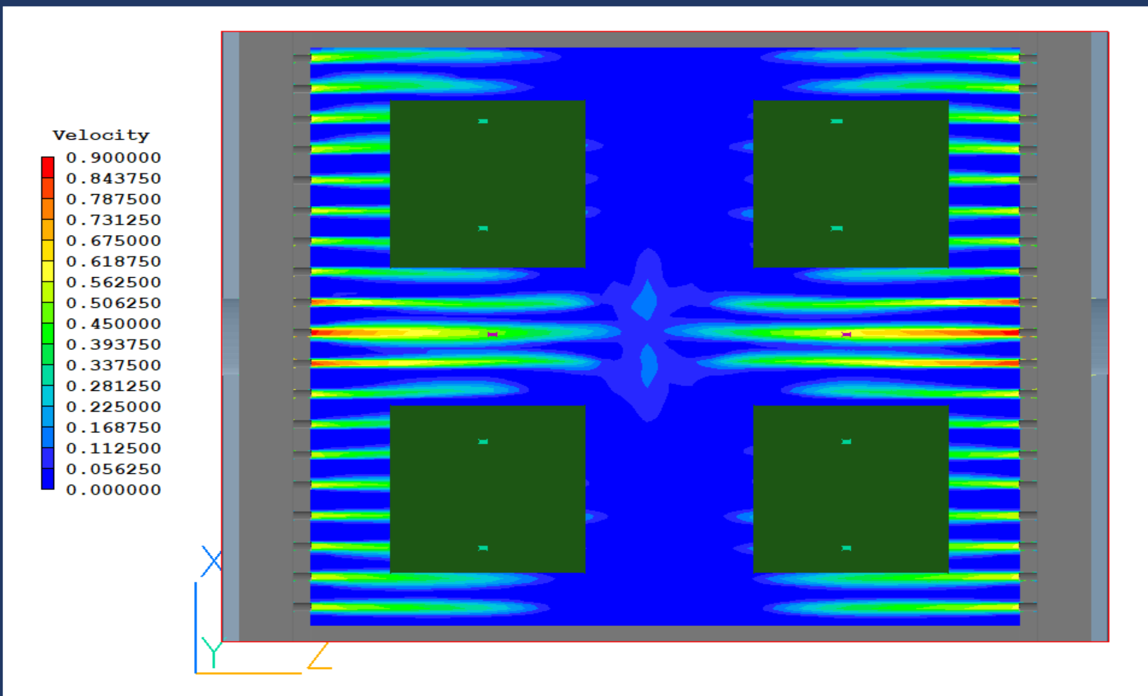


Figure 4.12b: Air mixing in the enclosure at the higher air flow rate - Top View

Ornamental Fountain

By Andrew Carmichael and Ryan Dyer of CHAM.

Water has been seen for centuries not only as a necessary part of life, but also as something that can be beautiful and calming, and powerful and terrifying. Ancient civilizations built stone basins to capture and hold precious drinking water as far back as 2000 BC and, since then, architects, designers, engineers and artisans have strived to increase the complexity and beauty of their creations, to deliver new experiences to their audiences. This has always been done through trial and error, which leads to a significant waste of time and materials when something goes wrong.

New technology, such as CFD, can minimize effort streamlining the design via numerous iterations to achieve just the right look for a fountain or basin before committing to constructing it.

With this in mind, RhinoCFD was used to simulate a (clearly-designed-by-an-engineer) water sculpture that showed the strengths of this design approach. RhinoCFD was used to apply the VOF method to two offset water sources converging in a central basin where they swirl and turn before curling down a slide to the bottom of the sculpture. The design was chosen to highlight key aspects that could be of use to artisans, such as the formation of static waves where the two sources meet, the swirling process itself and how this can be improved by altering the shape of the bowl, as well as the potential spillage of the water while descending the chute, indicating that design changes might be necessary to achieve a perfect basin.

The simulation was run for 9000 time-steps, each lasting 0.001s to give a total simulation time of 9s. Spatial discretization yielded a mesh of 125k cells and was computed running the simulation on a single processor for a total runtime of some 2h.

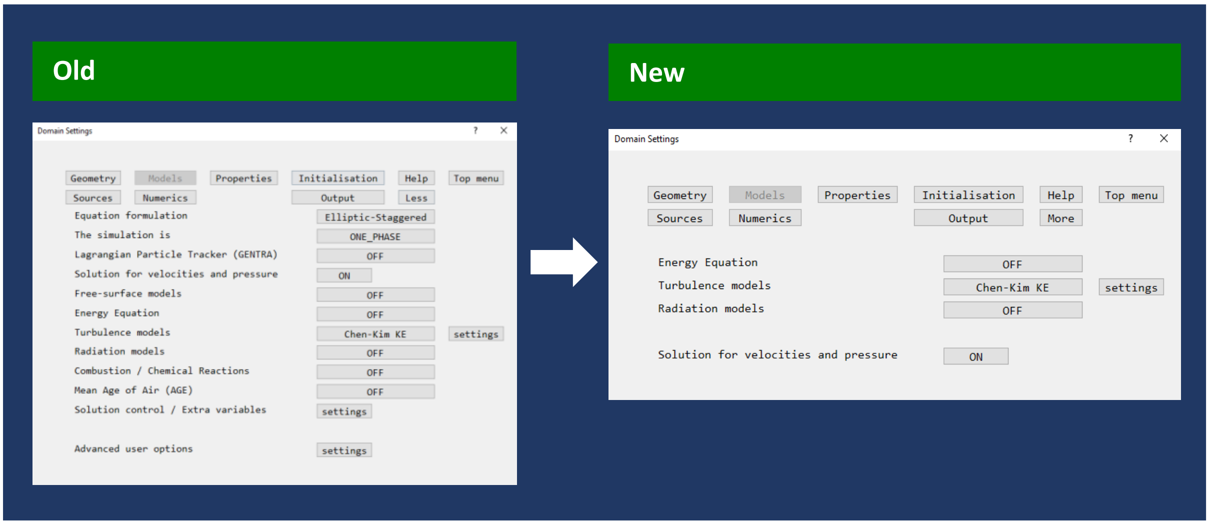
Thanks for hanging in there with us! We’ve been working to include many of your suggestions and advice into a full new version of *RhinoCFD.* Here are the major improvements we’ve made to the software so far:

Rhino 6 Compatibility

All the features will work with the same ease as you are used to in the past, while enabling many of the new features of Rhino6 to be compatible with *RhinoCFD*.

Simplified Menus

One of the key goals of *RhinoCFD* is to bring CFD to a host of new users that, typically, are not familiar with fluid simulations. With that in mind, we have further simplified our Main Menu (and updated the font) so that you are shown only the most relevant options needed to perform a simulation. The previous, full, Menus are retained behind a “more” button in case of need.



MiniRes

CHAM

Convergence information for run completed at 16:56:09 on Friday, 21 September 2018

on sweep 48

Title: 4 P

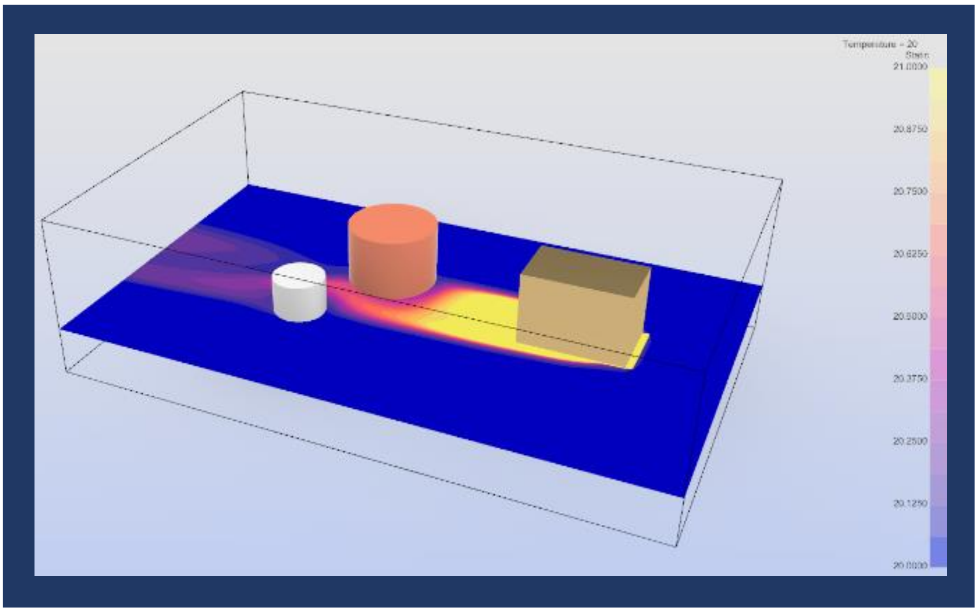
Variable	Source Sum	Residual Error (%)	Max Correction	Location (IX,IY,IZ)
P1 (Continuity)	-7.194234E-06 (kg/s)	0.997329	14.338472 (Pa)	(19, 52, 35)
U1 (X Velocity)	N/A	4.466197	0.425517 (m/s)	(39, 11, 9)
V1 (Y Velocity)	N/A	4.051003	0.732193 (m/s)	(41, 11, 8)
W1 (Z Velocity)	N/A	4.020025	0.260834 (m/s)	(18, 21, 20)
KE (Turbulent Kinetic Energy)	N/A	7.671095	0.166598 (J/kg)	(19, 22, 22)
EP (Rate of Dissipation of KE)	N/A	7.475048	431.278168 (J/kg/s)	(22, 33, 21)
TEM1 (Temperature)	4983.80957 (W)	9.718517E-10	124.245735 (°C)	(11, 36, 10)

CPU time of run 25 s

One of the key problems with CFD is that it is not always straightforward to determine whether your simulation has converged (worked) or not, and, hence, if you can trust the results produced. To assist with this, after every simulation, we have created a MiniRes (ult) file which displays any error in each equation solved. More importantly, it colour-codes the errors so you know what sort of levels *should* be acceptable for your run. Whilst not providing a guarantee, this should give a good idea of what is occurring.

Colour Maps

Rainbow colourmaps have always been popular for displaying data but recent research has uncovered several problems with them. How bright a colour is can affect how it is perceived and, because rainbow colourmaps do not typically control for brightness, they often include perceptual flat spots (where the change in colour is hard to distinguish) and perceptual bright spots (where the colour appears to change more quickly than it should). An ideal colourmap is one where the perception of the change of colour is the same throughout. These perceptually uniform colourmaps give a more accurate representation of the underlying data, and will work even for colour-blind users. This latest version of *RhinoCFD* contains a number of perceptually uniform colourmaps as well as traditional rainbow colourmap. Those included will be particularly useful for heating and particle dispersion simulations.



Looking to the future

Future planned developments include:

- Improved and simplified results panel and probe display
- PHISUM: combine wind results to obtain probabilistic wind conditions for pedestrian comfort
- Faster surface contouring
- Improved solar radiation modelling
- Faster solver time
- Wave Inlets

Analysis of Flow around a Ship Propeller Using the Rotor Model in PHOENICS CFD Software
By Dr. Kadir Özdemir. Simutek, Turkey

The purpose of this study is to calculate the force and torque values of a ship's propeller using the rotor model in PHOENICS CFD software. The propeller geometry was created as a three-dimensional solid using Rhino3D CAD software as in Figure 1.

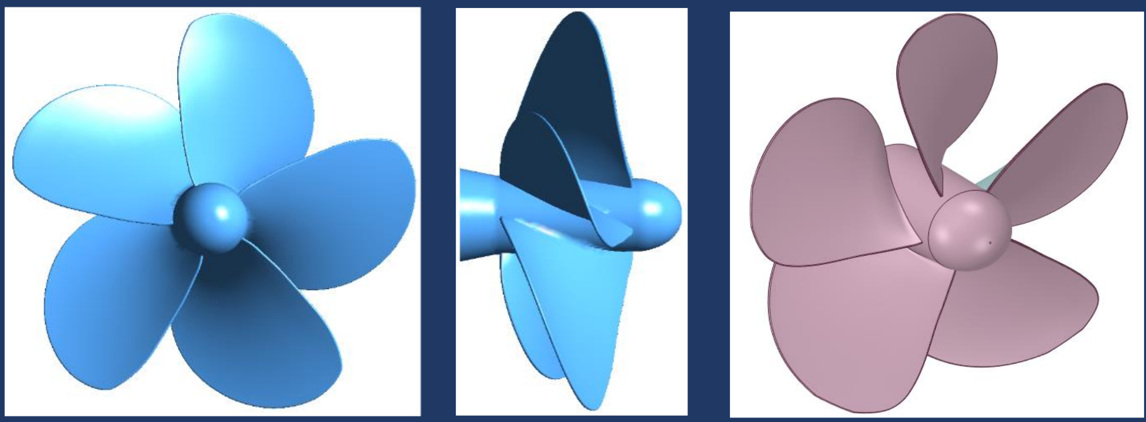


Figure 1. Propeller CAD Geometry

CFD Model

A cylindrical-polar solution domain was selected in PHOENICS, as shown in Figure 2 together with the propeller, propeller shaft and ROTOR object. The latter is a useful PHOENICS tool for simulating rotating objects like propellers, pumps and fans; it represents a zone of rotating co-ordinates in a cylindrical-polar grid, as shown in Figure 3.

Rotating co-ordinates are activated so that the propeller within the ROTOR can move relative to other stationary objects. The turbulence model used the rotational speed of propeller and the inlet velocity as given in Table 1.

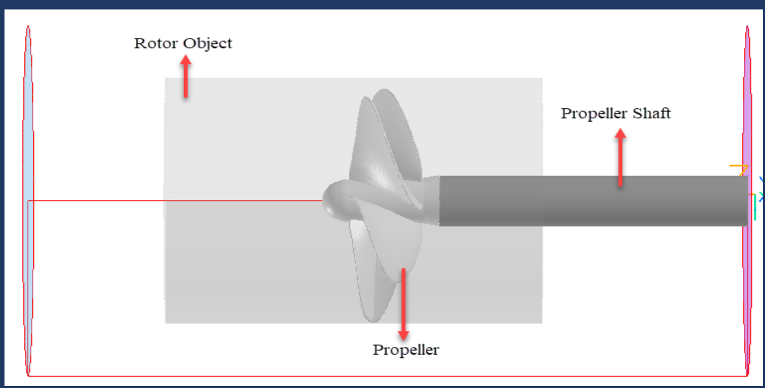


Figure 2

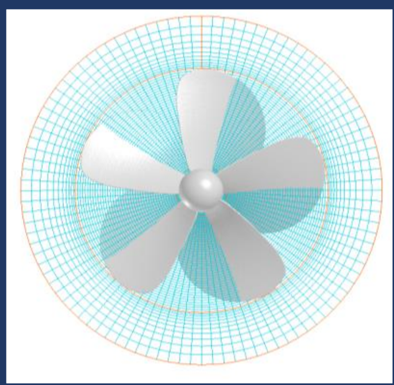


Figure 3. Grid

Table 1

Turbulence Model	Standard k-ϵ
Rotational Velocity	500RPM – 550RPM – 600RPM – 620RPM – 660RPM
Inlet Velocity	14,66 m/s (12 Knot)
Material	Water – Density 1025 kg/m3

Thrust and Torque results can be seen for different rotational speeds of the propeller in Table 2.

Table 2: Thrust and Torque

Rotational Velocity (RPM)	Torque (N.m)	Thrust (N)
500	1323	-20407
550	-2074	-4685
600	-5673	12037
620	-7180	19105
660	10332	33971

Results

Torque and Thrust values of the propeller were calculated at five different rotational speeds. Velocity and Pressure distributions can be seen in Figures 4 to 7 inclusive. Maximum velocity is 23 m/sec around the propeller at 500 RPM.

A video of the simulation can be seen on the SimuTek Engineering website or YouTube channel using the following links:

http://www.simutek.com.tr/tr/page/details/phoenics

https://www.youtube.com/watch?v=NGoO4p3gSG8

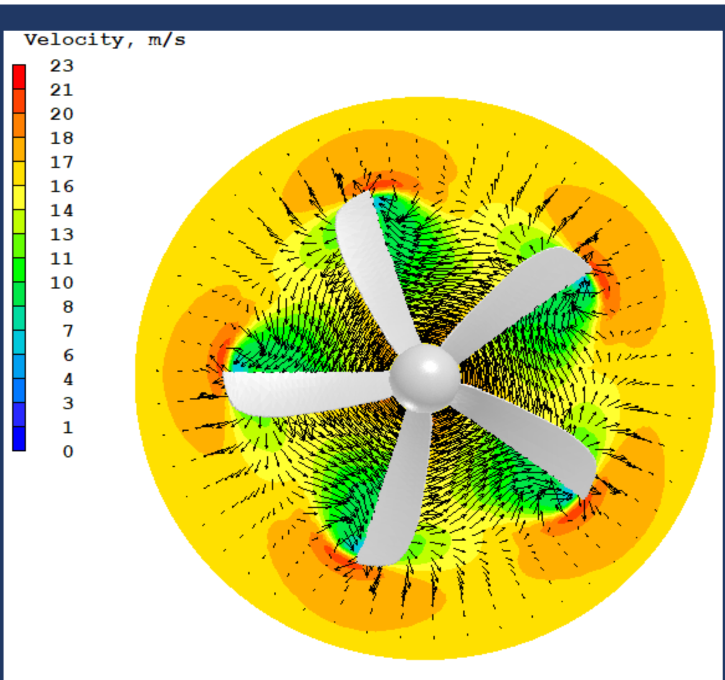


Figure 4. Velocity distribution around the propeller

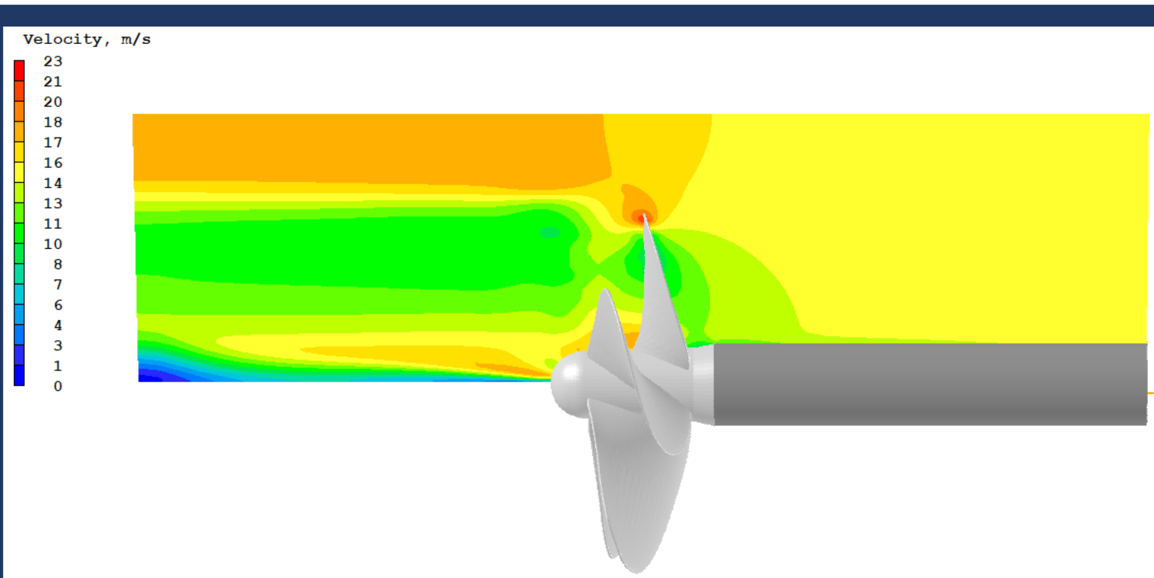


Figure 5. Velocity distribution around the propeller at section X

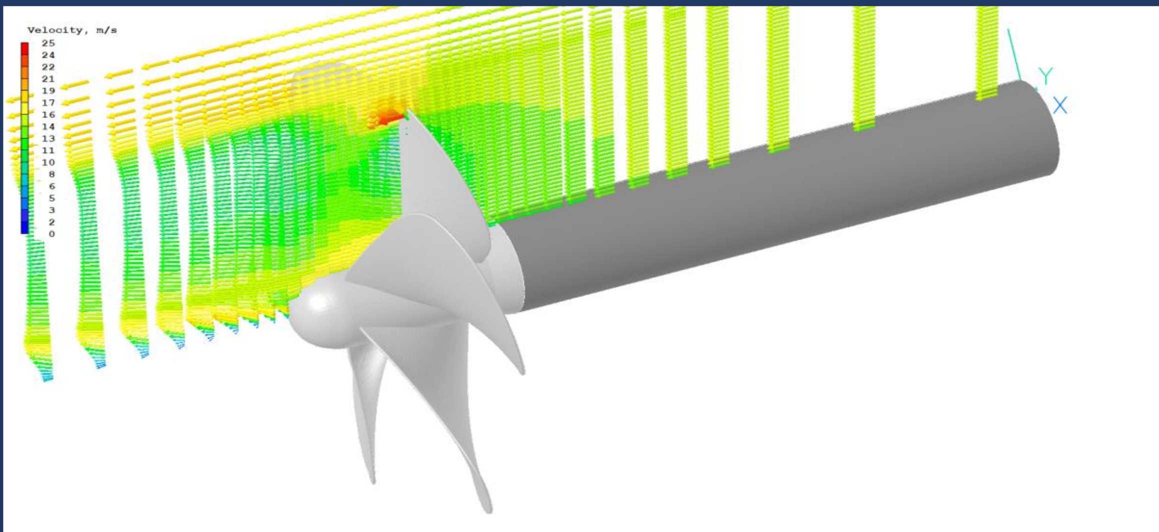


Figure 6. Velocity vectors at section X

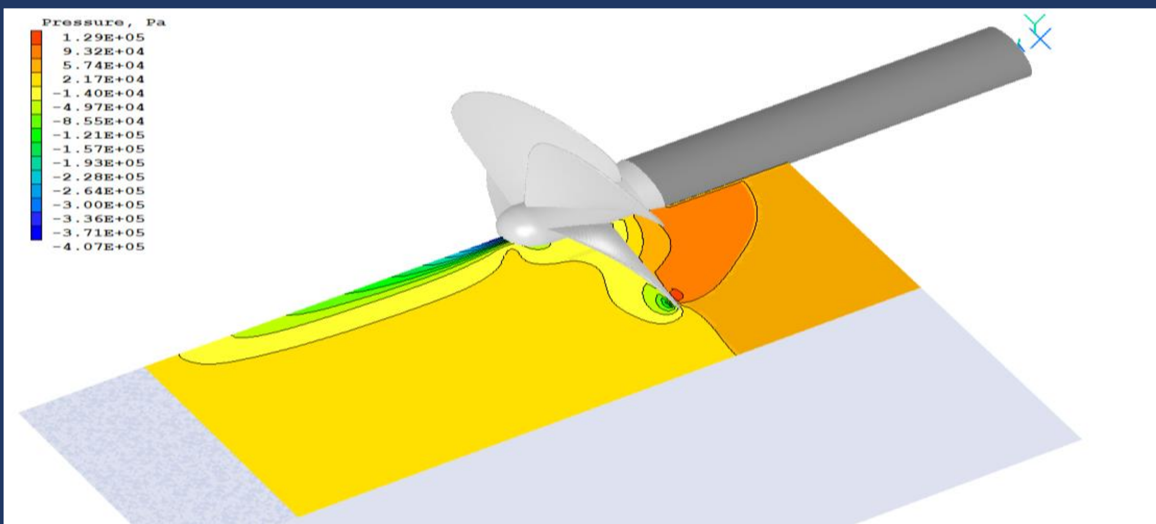


Figure 7. Pressure Distribution

CHAM Japan

Images from the User Meeting held at CHAM Japan on October 19 2018.

News from CHAM Agents

More than 85 participants from 17 countries were represented at the IMA9 Conference held in Guilin, China from August 31 to September 5 2018. Oral sessions and poster presentations, based on the topics of Marangoni and interfacial flows, were presented.

The work done on Marangoni and interfacial flows using the VOF methods of PHOENICS were presented by Dr Jalil Ouazzani from Arcofluid Consulting LLC (Orlando, USA). The presentation was well received and has led to many contacts.

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.