CHAM

Winter 2010

PHOENICS News

PHOENICS – your Gateway to Successful CFD

Happy New Year

Editorial

We wish all those reading this Newsletter a Happy New Year.

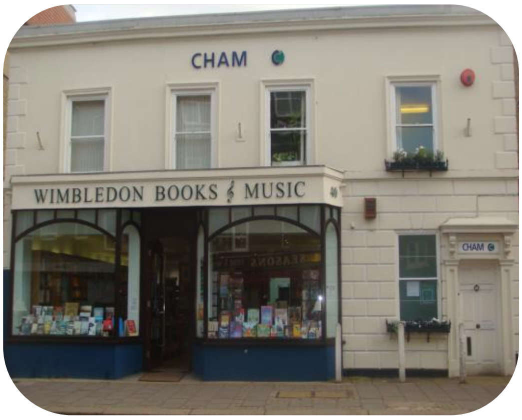
2010 has been an interesting year, financially, on an international basis and we are pleased that CHAM is managing to maintain, and improve, its performance despite the current recession.

This is due to the efforts of staff at Head Office in Wimbledon, in the CHAM Japan branch in Tokyo and at our Agents worldwide. We much appreciate these efforts and would like to thank all members of the international CHAM Team working to promote, and use, PHOENICS.

CHAM has increased the amount of consultancy work undertaken so we are extending the team carrying out this work and will have three new Engineers joining us in London.

CHAM Japan has also taken on staff to accommodate increasing sales.

We look forward to the continuation of this positive trend in CHAM in 2011 and hope it is a successful year for all PHOENICS Users.



CHAM Head Office, Wimbledon Village, London

Points of interest:

- CFD & PHOENICS
- Temperature-Dependent Specific Heats
- PHOENICS & Suspension Fired Vertical Cyclone Combustion Furnace
- PHOENICS in the Design of a Foam Separator for the Manufacture of Human Vaccines
- F1-VWT & PHOENICS
- PHOENICS & Dissipation Characteristics of Perforated Plates

Inside this Issue:

2) CFD & PHOENICS

2.1 Computer Simulation of Fluid Flow, Heat Transfer and Combustion: Can it be Trusted? *by Brian Spalding, CHAM Limited*

Abstract of Lecture delivered at ZAO 'Turbocon', Kaluga, Russia, December 9 2010

The lecture is addressed not to CFD specialists but to the engineering-equipment designers who rely upon their predictions of performance. Its main argument is that, although the scientific basis of CFD is sound, its practice rests also on the 'discretization hypothesis', namely the hypothesis that when laws are applied to a contiguous set of discrete space-time volumes, and these volumes are made successively smaller and more numerous, their quantitative outcomes will closely simulate reality, as illustrated by the sequence of images in Fig.1.

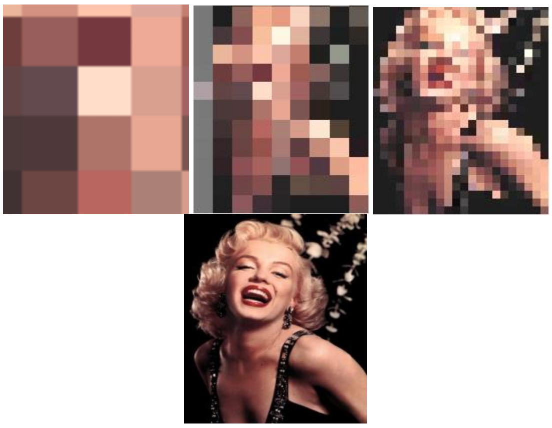


Figure 1 How successive grid refinement improves the closeness to reality.

It is the still-inadequate size and speed of today's computers which force CFD practitioners to use sub-divisions of space which are far too coarse to represent reality faithfully, for which inadequacies they seek to compensate by various stratagems, namely:

1. Use of detailed-geometry CFD only for small parts of the equipment to be represented, coupled with the use of space-averaged CFD for the equipment as a whole.
2. Use of turbulence models for the (very common) cases where turbulent-eddy sizes are below grid-cell sizes.
3. Use of multi-phase-flow models for cases with droplets, bubbles and particles of less-than-grid-cell size.

The particle-tracking method for multi-phase models is presented as an early population-modelling procedure which however becomes intractable when particle groups collide and merge. Discretized-population modelling is presented as its more practicable successor and illustrated by reference to turbulent combustion.

Primitive combustion models such as 'no-fluctuations', 'mixed-is-burned', 'eddy-break-up', 'presumed-pdf' and 'two-fluid' are represented on the TriMix diagram as single points or point pairs (Fig.2) ; and it is argued that only multi-member populations can do justice to the complexity of real flames. It is argued that the multi-fluid model, which allows selective discretization of the TriMix plane, provides an affordable means of introducing them, whereas Monte-Carlo-based 'pdf-transport' methods to not.

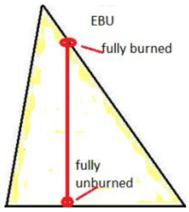


Figure 2 Eddy-break-up on the TriMix diagram

Engineering-equipment designers are advised to ask their CFD specialists to explain and justify their choices of model; but it is also argued that the ease of use afforded by packages such as PHOENICS Gateways enables designers to make the simulations for themselves.

2.2 The Specification of Temperature-Dependent Specific Heats by Michael Malin, CHAM Limited

PHOENICS can handle energy conservation in two alternative ways, namely: (a) by solving for enthalpy and deriving temperature from it; or (b) by solving for temperature directly, which is advantageous for conjugate-heat-transfer problems because the heat transfer between fluid and solid depends on differences of temperature and not those of enthalpy.

There has been some confusion among PHOENICS users when specifying a temperature-dependent specific heat for use in the energy equation based on temperature, mainly because PHOENICS uses an unconventional definition of the specific heat. The purpose of this article is to clarify this issue, so as to try and avoid any further implementation problems.

The energy conservation equation can be written as:

$$\frac{\partial}{\partial t}(\rho h)+\nabla.(\rho U h)=\nabla.(k_e \nabla T)+S_T \qquad (1)$$

where ρ is the fluid density, U is the velocity vector, T is the absolute temperature, k_e is the effective thermal conductivity, S_T is the source term, and h is the specific enthalpy defined by:

$$h=\int_{T_0}^{T} C_p dT \qquad (2)$$

wherein the specific heat C_p is the thermodynamic specific heat, i.e. it is the differential rate of variation of enthalpy with temperature at constant pressure, and T_0 is a given reference temperature.

For conjugate-heat transfer problems, PHOENICS uses the following form of equation(1) with temperature as dependent variable:

$$\frac{\partial}{\partial t}(\rho C_{p,e} T)+\nabla.(\rho U C_{p,e} T)=\nabla.(k_e \nabla T)+S_T \qquad (3)$$

where $C_{p,e}$ is the "effective" specific heat defined by:

$$C_{p,e} = \frac{h}{T} = \frac{\int_{T_0}^{T} C_p dT}{T} \qquad (4)$$

This **"**effective" specific heat is defined as the enthalpy of the material at the prevailing temperature divided by the absolute temperature. The enthalpy h may be expressed relative to a given datum at temperature T_0. It is important to note that this differs from the conventional "thermodynamic" specific heat at constant pressure, Cp, which is defined as the rate of change of enthalpy with temperature; but it has the same dimensions, and is of the same order of magnitude. This difference of definition between $C_{p,e}$ and C_p is important only when specific heat varies with temperature.

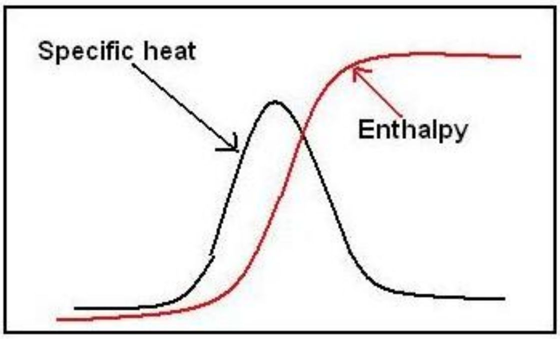
The effective specific heat can be set in the VR Editor from the "Main Menu~Properties" panel by selecting one of the built-in options from the pull-down list of available options, which are listed in terms of formula for C_p. The built-in options are somewhat limited, but if for example you want to set the specific heat to:

$$C_p = A + BT + CT^2 \qquad (5)$$

you can simply select the option "Quadratic in Tabs". More complex expressions for C_p can be set using In-Form. As a simple example, suppose that you want to implement equation (5) for C_p, then after making use of equation (4) to deduce $C_{p,e}$, the following InForm coding is needed in the Q1 file

SAVE9BEGIN
(stored of TABS is TEM1+273.0)
REAL(ACP,BCP,CCP);ACP=917.0;BCP=0.258;CCP=3.98E-5
*** Effective specific heat for air - 280K <TABS < 1500K*
(property cp1 is
:ACP:+0.5:BCP*TABS+0.3333*:CCP:*TABS^2)*
SAVE9END

More complex situations include flow and heat-transfer applications at supercritical pressures, which involve strong variations of fluid properties in the vicinity of the pseudo-critical point, as shown for example in the figure below:



Temperature increases to the right

For such applications, the specific-heat data are generally defined by tabulated values of enthalpy against temperature, and InForm can be used to great effect to compute $C_{p,e}$ from equation (4), after reading the enthalpy-temperature data from a user-defined external text file by means of InForm's piecewise-linear function. Sample InForm coding is given below:

SAVE9BEGIN
*** printout enthalpy implied by PHOENICS CP1 & TEM1*
*(STORED var ENT1 is (TEM1+273.0)*CP1)*
char(vn); vn = TEM1
*** Enthalpy in J/kg (piecewise linear from file ENTPRP)*
*(stored LFH1 is PWLF(ENTPRP,:VN:)*1000.)*
*** Effective specific heat - supercritical water at 24 MPa*
(property cp1 is LFH1/(TEM1+273.0))
SAVE9END

Finally, the 'Property storage' button in the VR "Main-Menu~Properties" panel allows field values of the effective specific heat to be placed in a 3D store named CP1, so that they will be available for printing in RESULT and for plotting with the VR Viewer from the PHI file.

3) PHOENICS Applications

3.1 Model of a Suspension Fired Vertical Cyclone Combustion Furnace using PHOENICS by Gerard M Carroll

(nb the diagrams have been selected from those submitted by the author and may not be the most representative of the article – ed)

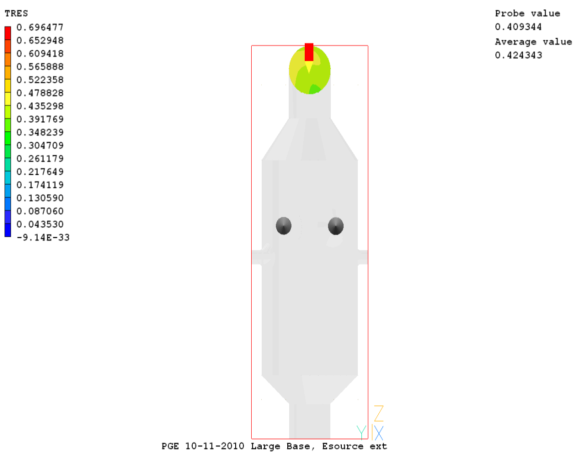
INTRODUCTION

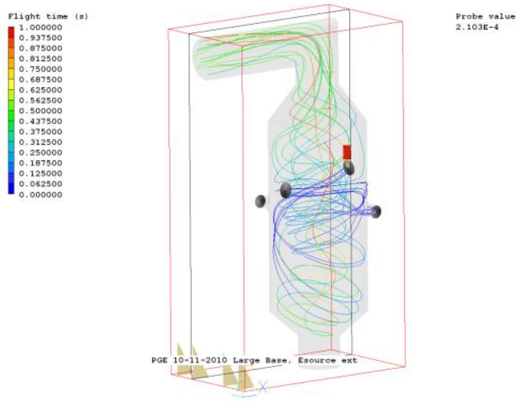
The PHOENICS model was constructed to determine the basic flow pattern in the furnace, and thereby to estimate residence time of the resultant gases.

The actual equipment comprises a vertical cylindrical furnace with two inclined pulverised fuel / air burners, and two preheater oil burners. The furnace is preheated to 1000°C using oil before the introduction of ambient combustion air and charcoal particles. Temperature is controlled by modulation of the air, as the plant must accept all the charcoal produced by the upstream process.

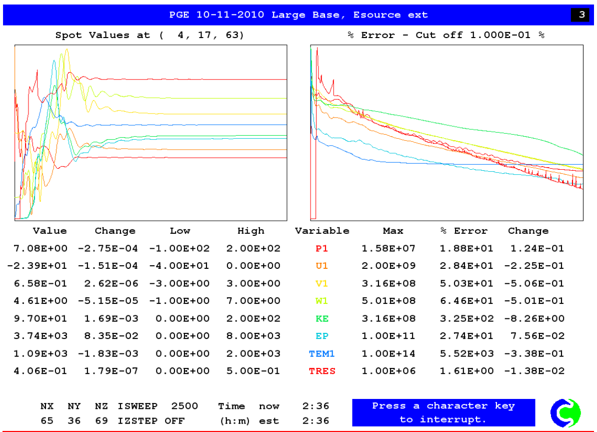
MODEL CONSTRUCTION

The model is currently a single gas model, using air. In other furnace models water vapour has been included to account for the water produced during combustion of hydrocarbons, but in this case burning dry pyrolysis derived charcoal, the level of hydrogen is very low and so a dry air model has been used.





With other incinerator residence time models, the gases have entered hot, as combustion has usually completed in the primary furnace. This model is effectively the primary furnace, and a method was needed to simulate the combustion heat release, without the complexity of a combustion model, particularly as the feed is solid fuel with variable particle size from dust to lumps of 10–15mm. Observations show that combustion in this type of furnace occurs in a belt around the outside wall. The approach taken was to create a heat releasing object occupying the approximate observed combustion volume. The heat release was set to match the theoretical heat release from combustion of the given rate of fuel feed.



RESULTS

The model converges very well in about 3000 sweeps and the flow patterns look realistic. Temperature profiles also look sensible, although there is a hot spot that would not be expected to be so extreme in practice. Real hot spots are observed, but the rather crude representation of heat release used should not be expected to predict these.

Residence time has been determined using both streamline time of flight, and cell by cell accumulation. In this model the results from both methods show a reasonable match. The cell by cell method can be a problem where there is a lot of mixing and recirculation, as this increases the amount of averaging that occurs, and may therefore mask the true minimum. It is intended to add a "numerical tracer" determination to the model where the converged model forms the basis of a transient model, solving only for tracer concentration over time. This method is probably the strictest test, and is appropriate where regulation requires a more accurate determination of minimum residence time.

Gerard M Carroll CEng CPhys MInstP MIMechE MIGEM
Director, AceFurnace Consulting Ltd
Tel 01494 784846, Mob 07875 036735

4) Consultancy Applications

4.1 Design of a Foam Separator for the Manufacture of Human Vaccines

by Paul Emmerson & Kate Taylor of CHAM Limited

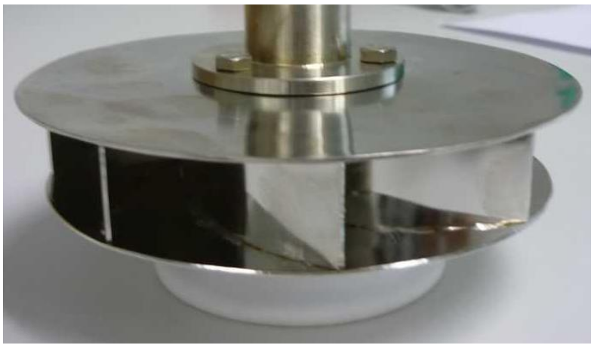
CHAM's consultancy team has recently undertaken a very interesting project in collaboration with Comberbach Consulting Ltd. (www.comberbachconsulting.com) to design an "induced-draft rotor". The rotor is used to separate (scavenge) foam from the surface of an aerobic culture of bacteria, growing in a fermentor.

Most stirred, aerobic microbial cultures generate unwanted foam that must be controlled to avoid blocking vent-gas filters. Most often, foam is destroyed with a mechanical foam breaker located in the fermentor head space, or controlled by repeat dosing of chemical antifoam into the liquid. However, in this case the foam contains one of the vaccine products, a fragile protein antigen that is destroyed by standard foam breaker designs. In addition, chemical antifoams are extremely toxic to bacterial growth and they interfere with the vaccine purification process.

In this study, CHAM was approached by Martin Comberbach on behalf of a large European fermentor manufacturing company, to design a foam breaker that would remove the foam from a bacterial culture without destroying the fragile vaccine product.

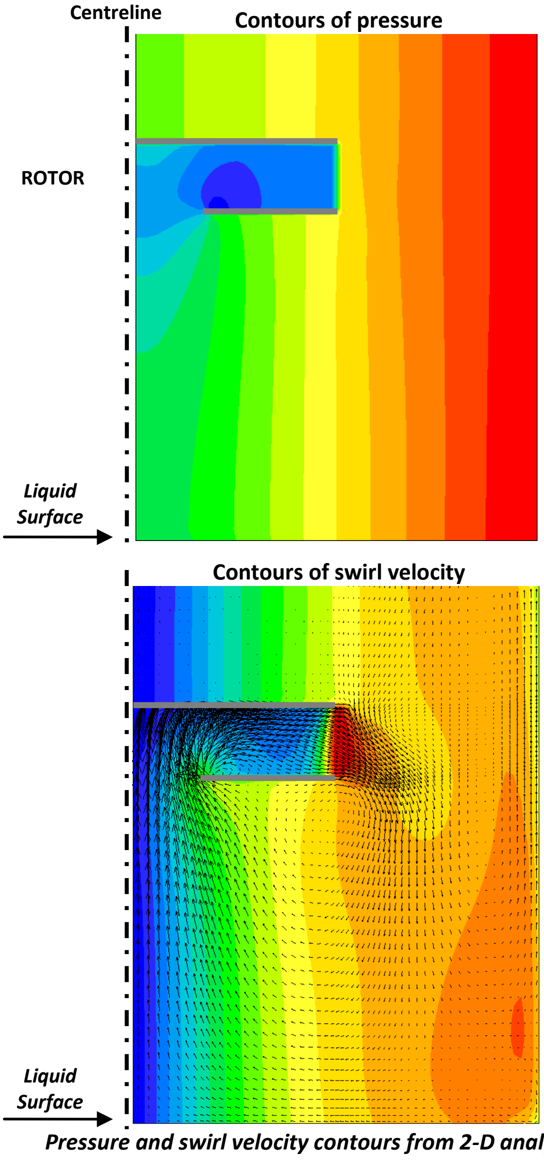
Foam reduction must involve a low-shear method and the best means of achieving this is an induced-draft rotor, commonly found in domestic vacuum cleaners. When a rotor of this design is rotated in the head space of a fermentor vessel, it creates a low pressure region, which sucks the foam from the liquid surface forming an 'inverted tornado.' The liquid separates from the gas inside the rotor and is thrown tangentially against the vertical fermentor wall, where it runs back into the bulk liquid under gravity. The rotor speed can be varied during the fermentation process, adapting the pumping force to the quantity of generated foam.

This is significantly different from conventional foam breaker designs in that the upper plate of the impeller is a solid disk, forcing the exhaust gas to flow upwards and outwards through the existing vent filter. Most conventional foam breaker designs are more complex, where exhaust gas exits through a hollow rotor shaft. In this case, CFD analysis was used to help optimise rotor design variables for maximum pumping capacity at the lowest rotational speeds. Rotors for 20 and 200 litre fermentor vessels were required.



Picture of a typical "induced-draft rotor" used for foam separation

The CFD analysis was performed in two stages, firstly a 2-D axi-symmetric (polar) model of the fermentor headspace in a static co-ordinate system, followed by a 3-D model of a single rotor blade passage using a body-fitted mesh in a rotating co-ordinate system. The 2-D model was used to simulate the flow induced in the vessel head space by the rotor and to predict the strength of the induced swirl, the up-flow velocity distribution below the inlet and the pressure rise across the rotor to pump at the prescribed volume flow rate. The purpose of the 3-D model was to predict the pressure rise a given rotor design can produce at a specified flow rate. The type of rotor considered here has a 'drooping' characteristic - i.e. the pressure rise which can be generated decreases with increasing flow rate - over most of its operating range. However if the flow rate decreases too far, or too high a pressure rise is demanded, the rotor can stall leading to a rapid loss of performance. The results of this model indicate whether the rotor is capable of pumping the required flow rate against the pressure difference generated by the induced swirl. Also the flow distribution in the rotor blade passage indicates the effectiveness and pumping efficiency for each rotor design. Both 2-D and 3-D models were run over a range of prescribed volume flow rates for each rotor design, with the flow rate vs. pressure rise relationship for each model plotted to find the intersection - thus giving the operating point or 'match point' for the rotor.



Pressure and swirl velocity contours from 2-D analysis

Several design parameters (e.g. blade height, number, angle, inlet and outlet diameter) were varied - within dimensional constraints imposed by the current vessel designs - in order to provide maximum suction at the liquid surface below the rotor for a minimum air volume flow rate. In general, the most powerful design parameters in terms of suction (or pressure rise) are rotational speed and tip diameter. Increasing the number of blades reduces the load per blade, and is likely to increase the rotor effectiveness across a large flow range. Changing the inlet hub diameter does not necessarily provide extra suction at the liquid surface itself, but it does change the operating point along the rotor characteristic. Once the main design parameters were set to give a reasonable overall rotor performance, fine tuning was performed to optimise the blade curvature, inlet and outlet angle. This is done to ensure the blade inlet angle is well matched to the angle of the inlet air flow, to avoid too high a positive or negative blade incidence; and the air is turned by the correct amount, as under-turning will result in too little pressure-rise, but over-turning might cause boundary layer separation and hence higher losses with less pressure rise.

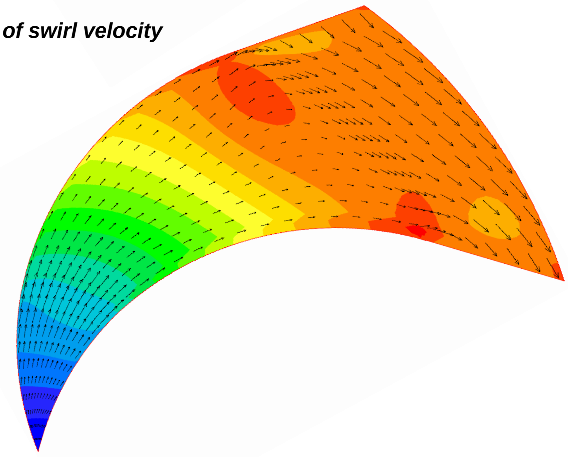
The effect of baffles (used to prevent vortexing in the liquid stirring process) extending into the vessel head space was investigated, and this was found to have a detrimental effect on the rotor performance. Also, the use of an inlet fairing to the rotor was investigated, and showed an improvement in the flow entering the rotor smoothly, reducing any early separation on the lower hub plate. However, the inlet fairing does move the rotor operating point along the characteristic, so for it to be beneficial overall, it is better to optimise the rotor design from the start including the fairing.

The design principles for aeration and mixing of liquid bacterial cultures in fermentor vessels have changed little over the last 50 years but the methods for control of foam have been largely empirical. Too often, fermentor manufacturers install ‘standard-design’ mechanical foam breakers at their clients’ request, without knowing whether they will work in practice. A CFD study of fluid flow in the fermentor headspace can give the client and fermentor manufacturer more confidence that its mechanical foam separator is fit for purpose.

In summary the project for Comberbach Consulting’s client was successfully concluded with two new rotor designs produced and analysed using PHOENICS, meeting the required specifications, and delivered ready for manufacturing.

Contours of pressure

Contours of swirl velocity



Pressure and swirl velocity contours from 3-D rotor analysis

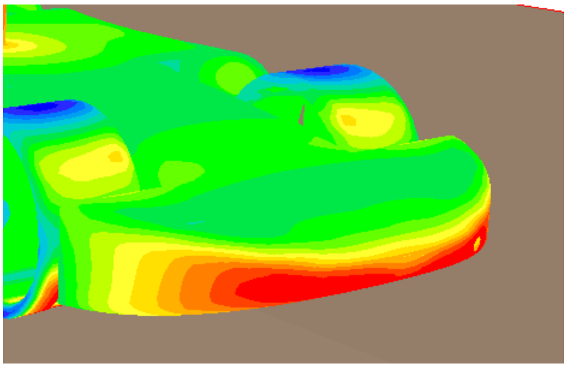
5) User Applications

5.1 F1-VWT Using PHOENICS

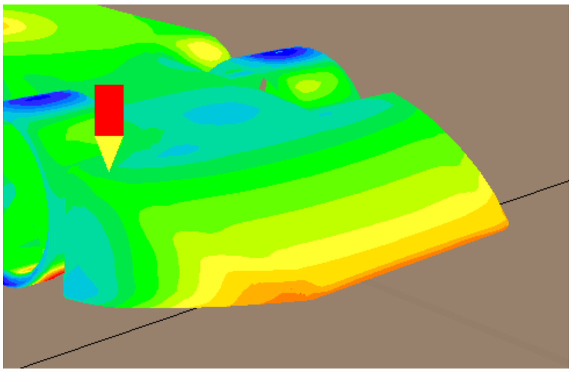
***by* Fred Stillwell, East Cobb Middle School STEM Associate Director Georgia BEST Robotics**

The F1 VWT software produced by CHAM played an integral role in the success of UNITUS Racing at the F1 in Schools world competition held in Singapore. The design engineers leaned heavily on this software to analyze many different design concepts to find the best design for the final car. This software was very intuitive and the designers were able to learn how to use it in only a few days. This allowed the team to spend more time designing virtually before they manufactured their final design.

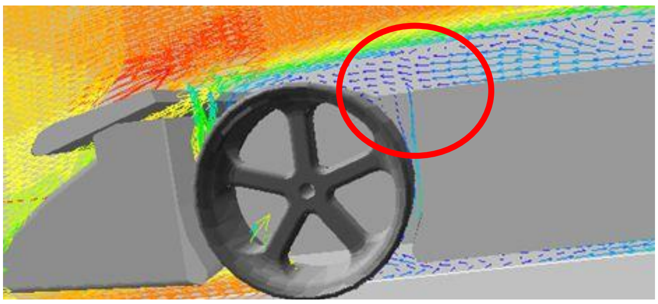
Before our team began to use F1 VWT the designers were creating the cars without knowing how the aerodynamics of the car was working. Some designs which were thought to be better were found to not work as well. The addition of this software really helped the team to make the final leap from competitor to champion.



The first image shows the front wing design that was used for multiple years before we noticed that it was not a great design. The designers knew that the first task would be to change the front wing design and the design below was found for the solution.



During the analyzation process the designers looked at the coefficient of drag in the X, Y, and Z directions. The Cdx value held a heavier weight to the designers because this is the aerodynamic drag that is affecting the car. This gave the designers quantitative data which were compared to other designs. Also, with the use of the VR Viewer the designers could look at the velocity vectors. The designers looked for signs of turbulence which were easily viewable using the F1 VWT package.



The above image shows the turbulence (circled) that the designers worked to eliminate through the design process.

Enabling designers to see how the air flowed around the different designs was extremely important because it allowed them to make very minute adjustments which made a big difference and this can all be done in a short amount of time. For our team it was very important because we only had 60 days to prepare for the competition and this software package allowed us to make the most efficient use of the time that we had.

The ability to show the judges visually the iterative changes that we made during the design process was extremely important because it showed that we understood the engineering behind our designs. This was integral during our engineering presentation where the judges were asking why we made a certain feature on the car and we could use the screen shots from the design process to back up our findings.

The F1 VWT package did everything that the team could have wished for and it made it feasible for the team to find the best possible design. The ease of the interface to learn and operate really made a huge difference for the team because it allowed them to easily learn and operate the software. In all, this software really did make the difference for UNITUS Racing and helped to put them on the top.

6) PHOENICS Use from Agents

6.1 Investigation of the Dissipation Characteristics of Perforated Plates using PHOENICS *by Stefano Malavasi & Gianandrea Messa, Politecnico di Milano, Milano, Italy*

Introduction

Perforated plates are widely used in pipeline systems either to reduce flow non uniformities by means of the correction of a distorted velocity profile or, placed side by side a valve, to attenuate the onset and the development of cavitation. The dependence of the pressure losses of these devices with respect to some of the most significant parameters – a very important issue when dealing with the cavitation phenomenon – is studied by means of numerical simulations performed with PHOENICS. The results are compared to experimental data reported in previous studies [3,5] and to the few formulas available in literature [1,2,6]. The work forms part of a paper [4] recently presented at the XXXII Italian Conference of Hydraulics and Hydraulic Constructions, held in Palermo between 14th and 17th September 2010.

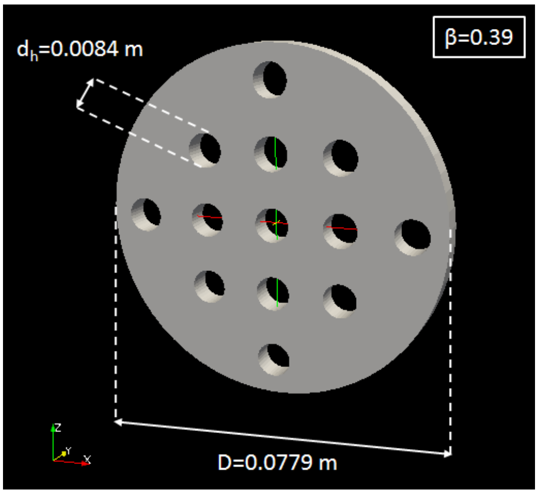


Figure 1 The plate considered in the simulations.

Computational model

The simulations concern the flow of water through plates of various thicknesses with the distribution of the holes reported in Figure 1, placed inside a pipe. The diameter ratio β, defined 2 as the square root of the ratio between the area occupied by the holes and that of the pipe, is equal to 0.39. The computational condition is given below.

1. Coordinate system is a 3D cylindrical system.
2. The boundary conditions are: a rectangular velocity profile (from 1.0 to 2.0 m/s) at the pipe inlet, with a turbulent intensity equal to 5%; an external pressure of 2 bar at the outlet; the generalized-log low at the walls.
3. Due to the geometrical symmetry of the device, and because the steady RANS are being solved, the domain covers one eight of the pipe section. Along the flow direction, 40*D* of straight pipe are simulated upstream the plate, in order to analyze the establishment of a fully-developed velocity profile. In the same way, 25*D* of straight pipe is modelled downstream the device in order to ensure the flow has reached the same fully-developed state.
4. The mesh, reported in Figure 2, consists of 40 tangential x 80 radial cells. Along the flow direction, the number of cells varies between 270 and 320 according to the thickness of the plate.
5. Differencing scheme is Harmonic Van Leer
6. Turbulence model is *k*-ε.

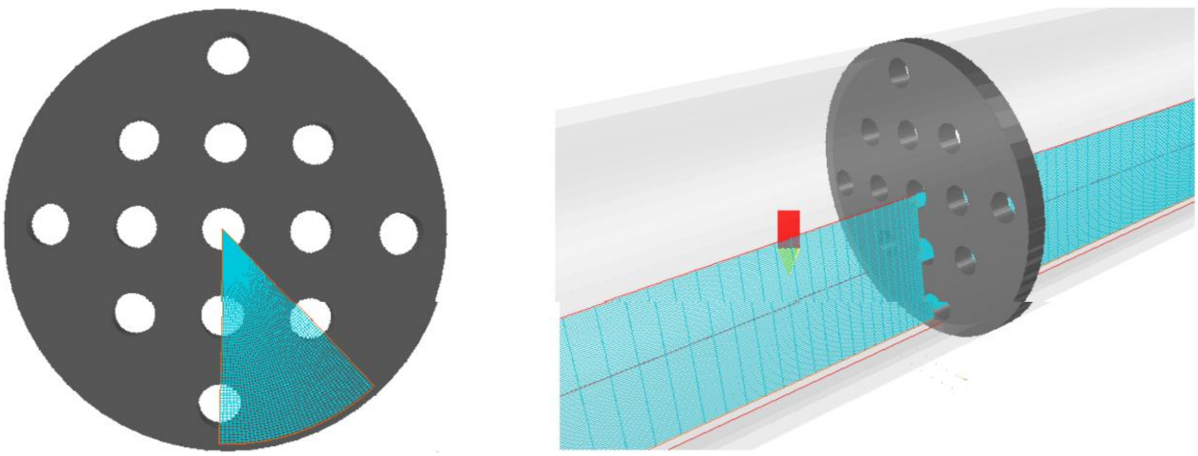


Figure 2 The cylindrical-polar mesh used in the simulations: planar view (left) and side view (right).

Results

The model is validated by means of a comparison to the experimental data, reported in previous works [3,5], concerning the case of relative thickness – i.e. the ratio between the thickness of the plate *t* and the diameter of the holes d_h - equal to 0.73. In particular, the comparison is made with respect to the pressure drop coefficient $\Pi_{\Delta\pi}$, defined as:

$$\Pi_{\Delta\rho} = \frac{\Delta\rho}{1/2\rho\,\varsigma^2}$$

where Δρ is the pressure drop across the plate, ρ is the fluid density and *V* the mean pipe velocity. The results, shown in Figure 3, appear in good agreement in the range of pipe Reynolds number considered, the deviation being lower than 8 percent.

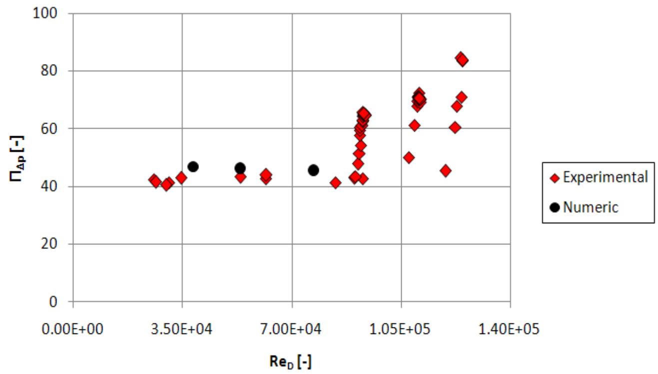


Figure 3 ***Trend of the pressure drop coefficient $\Pi_{\Delta p}$ as a function of the pipe Reynolds number R_D for the plate shown in Figure 1 (β=0.39) with t/d_h=0.73: comparison among experimental data and numerical predictions.***

The dependence of the pressure drop coefficient $\Pi_{\Delta\pi}$ upon the relative thickness t/d_h is then studied in the range 0.06 to 1.45, and the results are reported in Figure 4 together with the curves of Idelcick [2] and Miller [6], derived for the single-hole case. Two variants of the curve of Miller are displayed: the former is the original version of the Author [6], the latter is a corrected version cited by Fratino [1]. Since the theoretical models tends to overestimate the losses, especially for the lowest values of t/d_h, it could be supposed that, at least for the value of the diameter ratio β considered, the number and the disposition of the holes have some effect on the losses. However, it can also be observed that the trend of $\Pi_{\Delta\pi}$ upon t/d_h is qualitative similar to that of the variant of the curve of Miller purposed by Fratino [1].

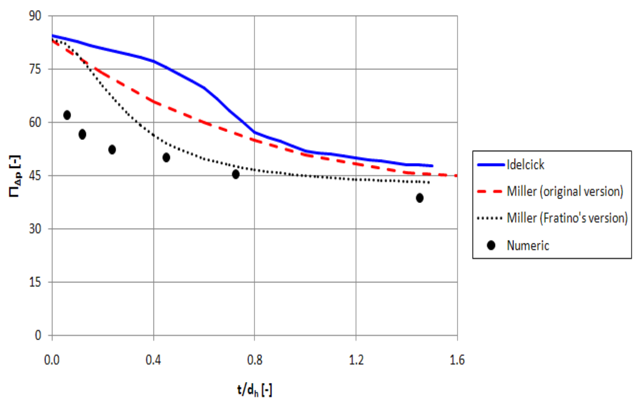


Figure 4 ***Trend of the pressure drop coefficient $\Pi_{\Delta p}$ as a function of the relative thickness t/d_h for the plate shown in Figure 1 (β=0.39): comparison among numerical predictions and literature models for the single-hole case.***

Conclusions

The dependence of the pressure drop coefficient through perforated plates with respect to some of the most significant parameters is studied by means of numerical simulations performed with PHOENICS. The results, in good agreement with the experimental evidence, show that, at least for the value of the diameter ratio considered, the number and the disposition of the holes, beyond the relative thickness, have some effect on the losses. As a consequence, the literature models developed for the single-hole case may not be suitable for the multi-hole one. The present study can thus be considered the first step of a more extensive research, aimed at developing a model to predict the pressure losses through perforated plates.

References

1. Fratino, U. Hydraulic and cavitation characteristics of multihole orifices, *Machinery and System - 20th IAHR Symposium*, Charlotte, NC, 2000.
2. Idelcick, I.E. Handbook of hydraulic resistance, 2nd ed, Hemisphere Publishing Corp, Washington, DC, 1986.
3. Malavasi, S., Macchi, S., & Mereghetti, E. Cavitation and dissipation efficiency of multihole orifices. *9th International Conference on Flow-Induced Vibrations FIV2008*, Prague, CZ, 2008.
4. Malavasi, S., Messa, G., & Macchi, S. The pressure drop coefficient through sharp-edged perforated plates. *XXXII Convegno Nazionale di Idraulica e Costruzioni Idrauliche*, Palermo, I, 2010.
5. Messa. G. Numerical investigation of the flow through orifices and perforated plates, Degree Thesis, Politecnico di Milano, Milano, I, 2009.

7) News and Events

7.1 Newsletter Contributions

If you would like to contribute an article to future issues of the Newsletter please email it to cik@cham.co.uk. Thank you.

7.2 PHOENICS Training Course

The next PHOENICS Training Course is scheduled to be held at CHAM in Wimbledon from January 19-21 2011. If you would like to attend please contact Peter Spalding on PHOENICS@cham.co.uk.

7.3 Staff

As per the editorial, three new members of staff will join CHAM in Wimbledon in January. CHAM-Japan has increased its staff by one. Information on our new colleagues will be included in the next Newsletter.

7.4 Eurostars

CHAM is continuing its collaboration on an EU-funded Eurostars Project in the field of wind energy, in conjunction with WindSim Norway and Iberdrola Spain.