CHAM
PHOENICS - your Gateway to CFD Success

PHOENICS News

Autumn / Winter 2013

Editorial

At ASME's 2013 Summer Heat Transfer Conference in Minneapolis in July Professor Spalding delivered a paper entitled *The Cyclic Pattern of Engineering Progress, with Lessons for Today* at one of the five sessions scheduled in honour of his 90th birthday.

The Evolution of Computational Heat Transfer Methods over the Past 75 Years, a panel discussion organized by A K Runchal and P Vanka, explored CFD/CHT developments. Panellists (B Spalding, S Patankar, M Wolfshtein, J McGuirk, and D Pepper below) reviewed key components of CHT and discussed questions from the audience.

Discussion Panelists, Minneapolis

Past students with Professor Spalding, Minneapolis

After the Dinner for Professor Spalding, Minneapolis

A celebratory dinner at a restaurant overlooking the City provided a most pleasurable gathering of students, colleagues and friends.

Another event marking Brian's 90th birthday, at Texas Tech on October 18, is intended to become the first of a series of meetings on Computational Fluid Dynamics and related topics.
At this event Professor Spalding's presentation was entitled *Past, Present and Future of CFD: A Limited Review.*

Mechanical Engineering Department, Texas Tech University

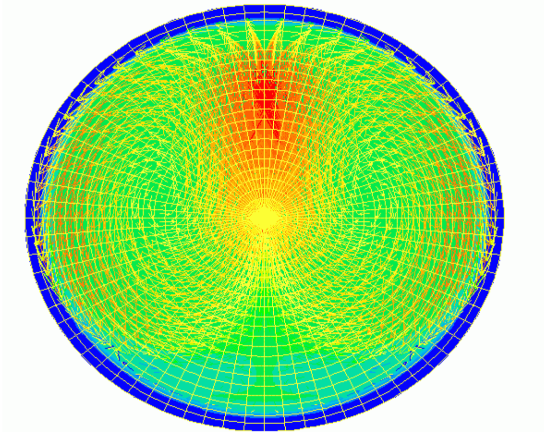
Past Students, October 18, Texas Tech University

Colleen King, Editor

Contents

1) CHAM Product Information

1.1 SimScenes



Velocity vectors & temperature contours in cooled tubes influenced by gravity acting in the vertical direction.

CHAM is working on a series of SimScenes - software packages enabling CHAM's **general**-purpose flow-simulation code, PHOENICS, to be used for **special** purposes by persons without PHOENICS experience.

PHOENICS possesses many built-in features and is able to simulate an extremely wide range of processes. It also has the capability to add new features and is equipped with sophisticated data-input and results-display facilities.

SimScenes are aimed at individual users with **particular simulation tasks** in mind who can call on a **sub-set** of the capabilities of PHOENICS via special Simulation Scenarios (SimScenes for short).

They have simple menu-type interfaces which address users in terms which they can reasonably be expected to understand. Pre-chosen default answers are displayed so users have only to make changes where it is desired so to do.

2) PHOENICS Applications

2.1 CFD Transient Modelling of Steel Billets in a Reheating Furnace by Rama devi Pathakota

Introduction

CHAM's consultancy team provided assistance to SABIC (Saudi Basic Industries Corporation) for the modelling of a reheating furnace. A reheating furnace is a facility used within steel mills for heating steel blocks (or billets) to the temperature required for rolling, producing end-products such as steel wire.

A typical reheating furnace contains three stages; 'pre-heating', 'heating' and 'soaking'. In the case shown below, the billets enter the furnace through a 'charging' door and then pass through the three stages to achieve their target temperature before exiting though a 'discharge' door onwards and towards the rolling process (not shown).

During pre-heating, the billets are warmed solely by the environmental temperature prevailing within the furnace. Direct heating occurs from above using gas burners to raise their temperature to the required level. Once achieved, the burners become less intense during the final soaking stage in order to retain an even temperature for each billet. The aim of this initial process is to heat up the steel billets uniformly to the optimum temperature, across both span and cross section, to make them malleable whilst avoiding billet breakage or damage to the machinery.

Reheating Furnace

A transient CFD analysis of the conditions operating within the furnace and heating process of the billets was performed to assist with the optimisation of the furnace design.

CFD model

Figure 1 shows a preliminary 2D CFD model setup, with the entry charging door, exit discharge door, steel billets and burners delivering, in the last two heating stages, hot gas at around 1200^{0}C. In this simulation the billets take one hour to transit across the furnace using a walking beam method to move them on, intermittently every 40 seconds. The 2D transient simulation was performed to predict the temperature rise and distribution across the billet cross-section after one hour in the furnace. The PHOENICS "INFORM" feature was used to set the initial temperature of the billet as it entered the furnace and to transfer the temperature from one billet to its next location when it was time for them to move on. This innovative approach involves a simple transference of the billet temperatures rather than modelling their physical movement, and is appropriate because the billets are all the same size and their transit time from one position to the next is very small compared to their stationary period.

The heat transfer processes modelled consist of (a) radiation between the furnace, its walls, burners and external links to the billets, and (b) convection from the hot gas. Radiative heat transfer between billets and the burners or air outside the charging door was modelled by setting an external link temperature using the "IMMERSOL" method available within PHOENICS.

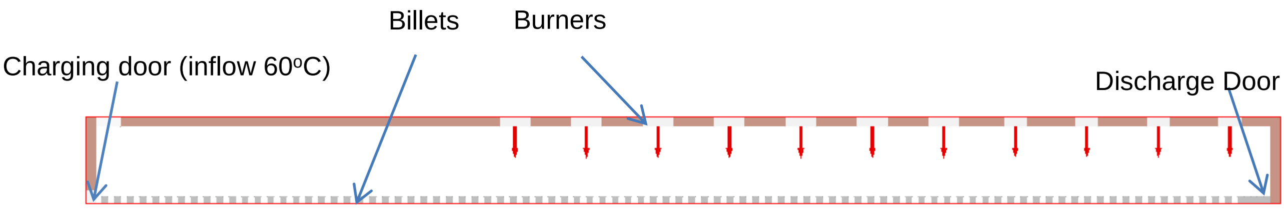


Figure 1 –2D transient CFD model showing charging door, discharge door, billets and burners

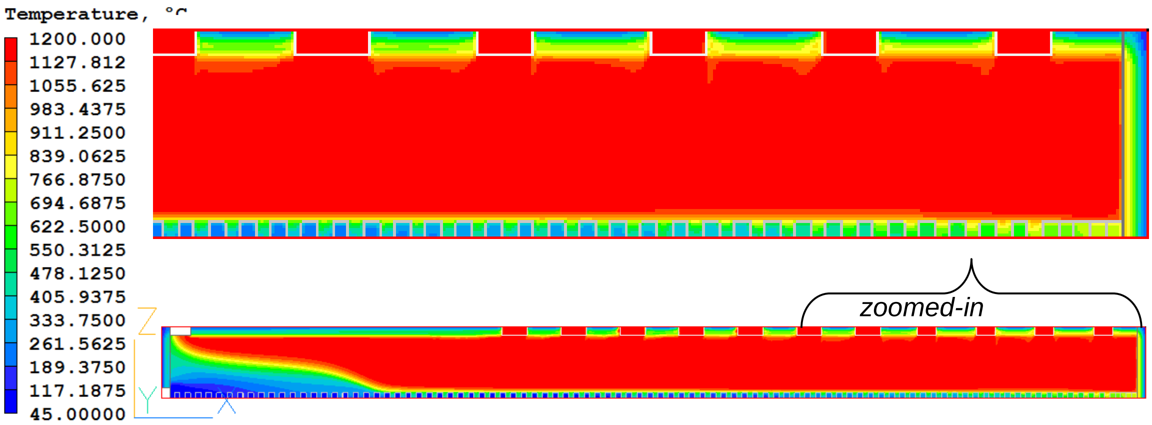


Figure 2 – Temperature distribution after 62.5 minutes

Results

Figure 2, shows the temperature distribution across the billet cross-section just over one hour inside the furnace. Relatively cooler air, at 60°C, enters the furnace through the charging door, along with each billet having a starting temperature of 45°C as it enters. The billet temperature gradually increases as it passes through the different heating stages, where burners are delivering hot gas at about 1200°C. The simulation predicts the temperature of a billet passing from the charging to discharge door to increase from 45°C to 750°C after one hour. Using the 2D model as its basis, CHAM helped SABIC engineers extend the case to construct a full 3D transient CFD model. This is now used to predict the billet temperature distribution in both cross-section and along the span, and represents the swirling flow created by the burners more accurately using PHOENICS FAN objects.

2.2 Vegetation Canopy Model for PHOENICS by Michael R Malin

Several companies have expressed an interest in using PHOENICS to simulate the interaction between the wind and forested areas, as well as with other types of tree or plant canopy, such as for example urban plantings of avenues or clumps of trees. PHOENICS has been equipped, therefore, with a vegetation canopy model to simulate the effects of vegetation by introducing flow resistance terms in the momentum equations, and turbulence production and destruction terms in the turbulence transport equations.

From an aerodynamic perspective, the main impact of vegetation on the environment is the reduction in air velocity due to drag forces, and the additional turbulence levels produced by the canopy elements. In PHOENICS these effects are represented by a porous-media approach based on superficial velocities where momentum sinks and turbulence sources are applied to a block of cells chosen to represent the tree canopy.

Flow resistance due to turbulent flow through the plant canopy is represented by a momentum sink term dependent on the superficial velocity vector, the drag coefficient, and the leaf area density (LAD) perpendicular to the flow direction. The default PHOENICS model uses a constant effective drag coefficient C_d and a constant effective leaf-area density α, although provision is made for the user to specify a vertical distribution of the leaf-area density. The canopy model also introduces additional source and sink terms into the transport equations for turbulent kinetic energy k and its dissipation rate ε. These terms account for turbulence production and accelerated turbulence dissipation within the canopy.

Momentum and turbulence source and sink terms were introduced into the PHOENICS finite-volume equations in linearised form by means of PIL and INFORM coding. More information will be provided regarding this model in a future Newsletter.

3) Agent News

3.1 ACADS-BSG: Australia, New Zealand, South East Asia

ACADS-BSG will exhibit at the ARBS Conference in Melbourne May 20-22 2014. ARBS is Australia's trade exhibition for the Air-conditioning, Refrigeration and Building Services industry. Showcasing the latest innovations and technologies ARBS connects local, national, and international exhibitors with major buyers, specifiers, design and consulting engineers and technicians. The event lasts 3 days and features a trade exhibition, a seminar program and the national awards gala event: www.arbs.com.au. The 2012 Exhibition attracted record crowds with 270 National and International Exhibitors displaying their wares.

ARBS 2012 Exhibition

ACADS-BSG Stand at the ARBS 2012 Exhibition

ARBS inducts eight to their Hall of Fame

At a Gala Presentation Dinner the winners of the esteemed ARBS 2012 Industry Awards were announced and eight nominees including Murray Mason, one of the Directors of ACADS-BSG, were inducted into the ARBS Hall of Fame, in recognition of their significant contribution to the industry.

Murray is a Life Member and Fellow of AIRAH, a Fellow of the Institution of Engineers and the Australian Institute of Energy, a member of the Australian Acoustic Society and a Director of ACAD-BSG Pty Limited, the Agents for PHOENICS in Australia and New Zealand. In 1989 he received a citation from AIRAH for his contribution to the development of the AIRAH Application Manual Series.

He has presented numerous papers at AIRAH Conferences and Seminars in addition to a number of international conferences and in 2003 won the W.R. Ahern award for the best paper published in the AIRAH Journal Ecolibrium. He has also been a sessional part time lecturer at VUT, University of NSW, Monash University and Swinburne University.

ACADS: 18 High Street, Glen Iris, Victoria
email: acadsbsg@ozemail.com.au
web: www.ozemail.com.au/~acadsbsg

3.2 ArcoFluid & ArcoFluid Consulting: France and the USA

Dr Jalil Ouazzani, now based in Orlando Florida, attended the 2013 ASME Summer Heat Transfer Conference at which he delivered a paper entitled *A Novel Numerical Approach for Low Mach Number – Application to Supercritical Fluids* in the Track 21 Symposium in honour of Professor Spalding.

USA: 390 North Orange Ave, Suite 2300, Orlando, FL
phone: 407 272 1928
email: jalil.ouazzani@gmail.com
web site: www.arcofluidconsulting.com

France: 3 Rue du Golf, 33700 Merignac
Phone: (33) 556 347 536
email: arcofluid@arcofluid.fr
web: www.arcofluid.fr

4. PHOENICS Applications: External Users

4.1 PHOENICS Modelling of Wind Turbines Using Moving Momentum Sources by Dr R P Hornby

Introduction

The United Kingdom (UK) is one of the best locations for wind power in the world, and is considered to be the best in Europe. The London Array is the largest offshore wind farm in the world with a capacity of 630 megawatts (MW) (figure 1), but this could soon be surpassed as proposals for a £3.5billion wind farm off the south coast of England (the Navitus Bay Wind Park, potentially delivering 900 MW) have been submitted to the UK government. If this wind park goes ahead, it would be located off the Dorset and Hampshire coasts, to the west of the Isle of Wight, involving up to 218, 200m high wind turbines. If approved, construction is expected to start in 2015.

Wind power delivers a growing fraction of the energy in the UK and at the beginning of January 2014, wind power in the UK consisted of 5,276 wind turbines with a total installed capacity of over 10 gigawatts (GW) comprising 6,831 MW of onshore capacity and 3,653 MW of offshore capacity. The UK is ranked as the world's sixth largest producer of wind power, having recently overtaken France and Italy. In 2012, 19.4 terawatt hours (TWh) of energy were generated by wind power, which contributed 5.3% of the UK's electricity requirement.

Figure 1. The London Array is an offshore wind farm in the Thames Estuary in the UK. With an installed capacity of 630 MW, it is the world's largest offshore wind farm.

The performance of a wind farm is influenced significantly by the state of the atmospheric boundary layer and wind turbine wake interactions (see figure 2 for a rather pronounced environmental effect). Computational Fluid Dynamics (CFD) is now being used to provide an integrated approach to modelling the detailed spatial variation of the wind field for both onshore and offshore wind farms (ref 1). In these models, the swept disc of the turbine rotor blades defines an area over which the flow perturbations induced by the turbines are set depending on the rotor performance characteristics (e.g. its thrust performance curve). This can be achieved by fixing the perturbations within the disc (ref 2) or assigning a porosity distribution to the disc which mimics the required thrust variation (e.g. the WindSim model, also based on PHOENICS, ref 3). However, a more direct approach, which is considered here, models the effect of each rotor using momentum sources attached to each moving rotor. These sources are based on the rotor lift and drag characteristics, thus allowing the rotors to be represented on a relatively coarse grid and additionally enabling calculation of the induced rotational fluid motion (which is not represented in the former two modelling approaches).

Figure 2. Horns Rev 1 is an 80 turbine wind farm off the western coast of Denmark, commissioned in 2002, and delivering 160MW. The above photograph shows the turbulence field downstream of the wind turbines. Unique meteorological conditions on 12 February 2008 at 1300 hours resulted in the wind turbines creating condensation (i.e. clouds) of the very humid air, thus making it possible to see the turbulence pattern behind the wind turbines.

PHOENICS modelling

A single 3 rotor wind turbine is modelled using a 3D Cartesian grid with *x* (lateral), *y* (vertical) and *z* (axial) coordinates. The grid is arranged with the finest discretisation surrounding the turbine. The 3 velocity components and the potential temperature are solved (using KOREN) together with the density which is obtained from the ideal gas law. The full buoyancy term is included in the vertical momentum equation. The standard k-ε turbulence model is employed, though with the constants adapted for use in the atmospheric boundary layer (ref 1, 4). The laminar Prandtl Number of heat is calculated in Ground in a way that allows variability in the use of the corresponding turbulent Prandtl Number. This also means that the production term for turbulence from buoyancy requires explicit coding in Ground.

On the low boundary inflow conditions are imposed using Monin-Obukhov (MO) type similarity functions (ref 2, 5). On the high boundary a pressure distribution derived from an assumed density variation with height is initially employed. At a specified time during the calculation, this pressure distribution is modified and fixed to correspond to the actual pressures calculated near the exit plane. This ensures a smooth outflow.

East, west, north and south boundaries are assumed solid and frictionless. On the south boundary velocities and turbulence quantities are fixed near the ground using the MO similarity functions. In order to represent different atmospheric stability conditions a heat flux can be imposed on the south boundary with a corresponding heat flux on the north boundary.

The perturbation to the flow produced by the wind turbine rotors is modelled using rotor lift and drag coefficients. The surface of each rotor is subdivided into a set of 2D panels each of which contributes a momentum source depending on the panel area and the local lift and drag coefficients. As the rotors revolve, these momentum sources are applied to the PHOENICS cell into which a particular panel falls. This produces a drag effect on the fluid as well as a swirling motion. Typically 50 time steps are used per rotor cycle. After a few cycles a quasi-steady state is reached. At the moment the rotor speed is specified but could form part of the calculation if the moment of inertia and the frictional couple of the turbine are known.

Figure 3 shows a typical contour plot of velocity magnitude in the plane of a 3 rotor turbine with in-plane velocity vectors superimposed. This illustrates the rotary fluid motion induced.

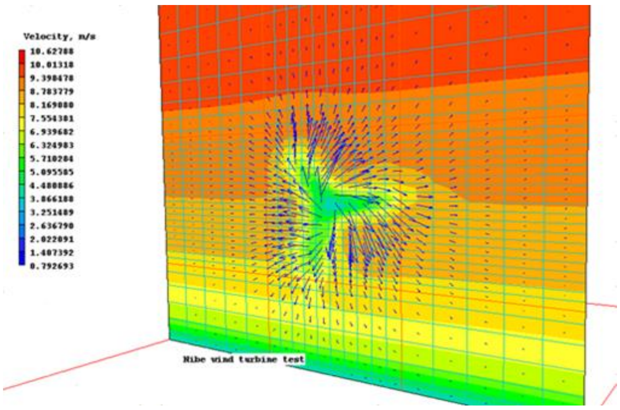


Figure 3. Velocity magnitude contour plot in the plane of a modelled turbine with superimposed in-plane velocity vectors.

Comparison of results with experiment

Results are compared with experiments undertaken in Denmark in 1990 on two Nibe wind turbines, 40m in diameter with a hub height of 45m and separated by 5 diameters. For this particular calculation, 20, 41 and 60 cells are used in the *x*, *y* and *z* directions in a region with dimensions of 120m by 200m by 800m respectively.

The atmospheric state is assumed to be neutral. Experimental results, surface roughness and friction velocity for input to the PHOENICS model are taken from ref 2.

A comparison with the PHOENICS results for the axial velocity is shown in figure 4. The surface roughness is set at 0.08m and the friction velocity at 0.56m/s. The upstream PHOENICS results agree very well with the similarity function (which for a neutral atmosphere is just a logarithmic variation). Reasonable agreement is also obtained with the experimental velocities at 1D and 2.5D downstream of the turbine.

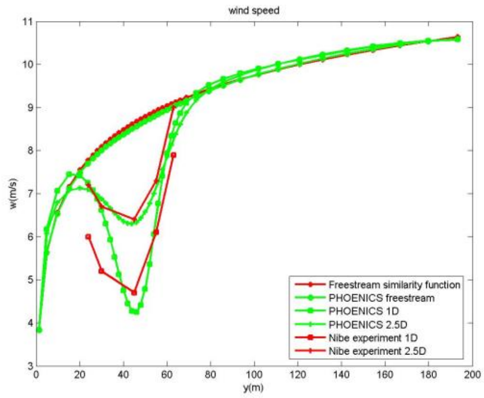


Figure 4. Comparison of PHOENICS results for the axial wind velocity with experimental results from the Nibe wind turbine experiments. The surface roughness is assumed to be 0.08m and the friction velocity 0.56m/s.

Turbulence intensities (derived in the PHOENICS results as the square root of two thirds of the turbulent kinetic energy) are shown compared at 2.5D downstream in figure 5. For this test the wind speed is lower and the friction velocity set at 0.46m/s. The agreement is satisfactory in establishing the viability of this method and provides encouragement for further extension and testing of this methodology.

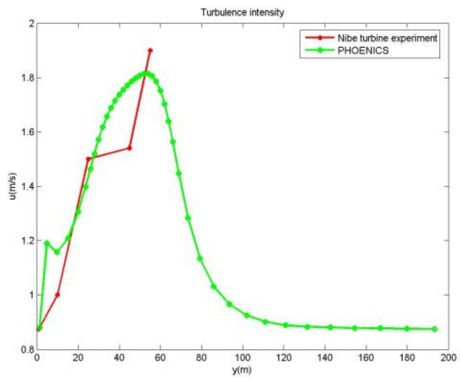


Figure 5. Comparison of PHOENICS results for the turbulence intensity with experimental results from the Nibe wind turbine experiments. The surface roughness is assumed to be 0.08m and the friction velocity 0.46m/s.

Conclusions

Motivation for the current work has been to provide a CFD wind turbine model that naturally incorporates the swirling flow produced by the turbine rotors and can be represented on a relatively coarse grid. Comparison of the model results with the (limited) experimental data available to the author shows encouraging agreement with axial velocity and turbulence intensity downstream of the turbine. However, no comparison of the model's capability in producing swirl has been carried out due to lack of available experimental data.

References

1. Barthelmie R J, Frandsen S T, Hansen K, Schepers J G, Rados K, Schlez W, Neubert A, Jensen L E, Neckelmann S. Modelling the Impact of Wakes on Power Output at Nysted and Horns Rev. EWEC '09 Proc., Marseille 2009.
2. Hernandez J, Crespo A. Wind Turbine Wakes in the Atmospheric Surface Layer. PHOENICS Journal Vol.3, No.3 1990.
3. Crasto G, Gravdahl A R, Castellani F, Piccioni E. Wake Modelling with the Actuator Disc Concept. Energy Procedia 24, 385-392, 2012
4. Prospathopoulos J M, Politis E S, Chaviaropoulos P K. Modelling Wind Turbines in Complex Terrain. EWEC '08 Proceedings, Brussels 2008.
5. Panofsky H and Dutton J. Atmospheric Turbulence. John Wiley & Sons 1984.

Dr R. P. Hornby e-mail: bob@hornby007.wanadoo.co.uk

5) CHAM News

5.1 CHAM Japan

CHAM Japan held a successful and well-attended Summer User meeting; some photographs are shown below. www.phoenics.co.jp

5.2 CHAM London

Three new Engineers joined CHAM at the Wimbledon Office in January. More information in the next Newsletter.

To contribute to this Newsletter please send articles, in Word format, to cik@cham.co.uk. Thank you.

6) PHOENICS Diary

2014	Activity
On going	*Shanghai Feiyi will be holding training courses for PHOENICS 2012 in other cities including Fuzhou and Nanchang. For full information please see www.shanghaifeiyi.cn.*
On going	*C-h-a-m-p-i-o-n, Taiwan, Provides basis and Advanced training on a regular basis. Contact: sales@c-h-a-m-p-i-o-n.com.tw or www.c-h-a-m-p-i-o-n.com.tw*
On going	*CHAM London holds regular PHOENICS training courses. Please see www.cham.co.uk*
Feb 2014	*Safe Solutions Brazil is one of the companies participating of the Accelerate Oil & Gas Business Roundtable in February 2014 in Rio de Janeiro. Several large national and international companies have the opportunity to access the Safe Solutions portfolio, providing the possibility of future partnerships. Among the confirmed "anchor" companies are some customers of Safe Solutions and/or CHAM such as Petrobras, Shell & Odebrecht. www.safesolutions.com.br/en/software-4/phoenics*
On going	*Focus Advance Technologies, Malaysia, provides training for beginners, intermediate and advanced CFD users. The training for beginners is usually for first time users with zero background in engineering. Intermediate training is for those with an engineering background who wanted to add new skills. Advanced user training is for those who use CFD regularly but require deeper understanding of the theory behind the CFD software. www.focus-technologies.com.my*
On going	*ACFDA, Canada, provides multi-level training to ensure that customers become knowledgeable users of CFD models and PHOENICS CFD software.* *Various training course options are available:* *1. Basic 3-day courses for new customers (includes free 1-month license & working template for client application)* *2. Advanced courses with in-depth coverage of specific PHOENICS features such as modelling multiphase flows, combustion, etc.* *3. Customized courses to suit particular client requirements.* *Training sessions are offered: At our Toronto office, on client sites, over the internet. For more information please contact info@acfda.org.*