PHOENICS News

Spring 2010

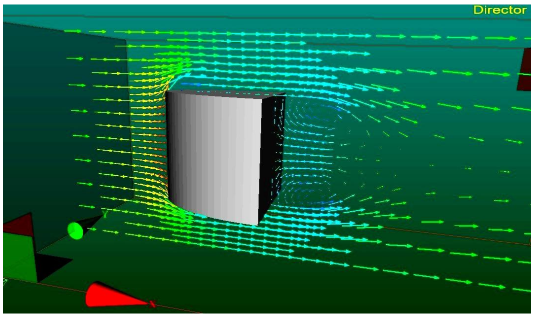
1) Editorial

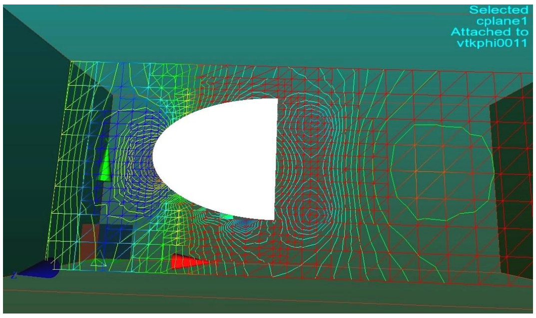
Progress with PRELUDE

Some PHOENICS users have preferred to use the VR interface rather than the PRELUDE pre-processor because the latter has been unable to display the results of calculations.

This has now changed; moreover PRELUDE can now display results obtained from both structured and unstructured grids whereas the VR-Viewer can handle only the former.

Below are two images from PRELUDE's Virtual Wind Tunnel Gateway showing grid, vectors, and contours. This is still work-in-progress, but progress is rapid having started only in mid-January 2010.





CONTENTS

2) PHOENICS Applications

2.1 Educational Use of PHOENICS in St Petersburg by Ekaterina E. Kitanina, PhD

1. Introduction

In the last few years PHOENICS has received growing recognition in Russian Universities. This popularity is encouraged both by user-interface simplicity and a wide range of physical and technical problems available for numerical simulation.

In Saint-Petersburg State Polytechnical University we use PHOENICS for teaching Masters Degree students at the Power Engineering Faculty. Indeed a modern engineer should have a comprehension of numerical simulation techniques and also be able to perform independent simulations. Of course the aforesaid doesn't mean that numerical simulation techniques could completely replace the experimental investigation experience for a student. Only a combination of both numerical and experimental investigation provides a deep understanding of physical effects and gives the necessary professional skill.

We developed a special course that includes experimental and numerical investigations and their comparison. The experimental part of course was developed under the direction of Prof. V. U. Mitiakov. The experiments are based on the Particle Image Velocimetry technique (PIV). The POLIS complex created by Novosibirsk Institute of Thermophysics SB RAS was used [1].

Classical aerodynamic problems such as flow around a circular cylinder and a boundary layer on a flat plate are considered. Below a two-dimensional air flow around a cylinder at Re = 500 is discussed as an example.

2. Flow around a cylinder: PHOENICS and PIV results obtained during the course

Flow around a circular cylinder generates a von Karman vortex street that can be observed both by numerical and experimental investigation. This classical hydrodynamic problem has a detailed description in literature [2, 3]. It gives an additional opportunity of comparison and analysis.

In our experiments air flow around an 18 mm diameter rod has been investigated. The flow was produced in a working part of a wind tunnel at the department of Thermodynamics and Heat Transfer. The velocity of air was equal to 0.42 m/s. The turbulence level didn't exceed 0.3%. The double cavity Nd:YAG laser with a pulse energy of 70 mJ was used for the illumination of small seeding particles carried by the flow.

The maximum frequency of velocity field measurements was equal to 3Hz. A digital camera with a 2048x2048 pixel resolution was used for image recording. Data processing was performed by ActualFlow software.

The numerical simulation was performed for a viscous laminar flow at Re = 500. Computations were fulfilled using non-uniform grids having up to 100000 cells. The value of a physical time-step providing sufficient time-accuracy was equal to 0.1 s.

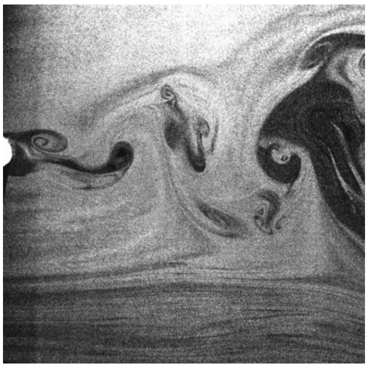


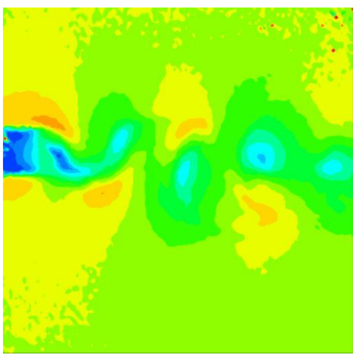
Fig. 1. A flow snap-shot obtained during the experiment.

A flow snapshot seen by laser illumination of the particles flowing around the cylinder is presented in Fig. 1. This photo, obtained by an initially high smoke particle concentration, gives an idea of the stream traces.

Velocity fields obtained by experimental and numerical investigations are shown in Fig. 2 (a, b). The von Karman vortex street can be clearly seen in both figures. Some discrepancies corresponding to the vortex shape and velocity level are explained by the intentional simplifications of the numerical simulation problem definition. These simplifications are due to the student course limits.

The numerical simulation was performed until the time variation of velocity value in monitoring cells became periodical and independent from initial conditions. Then the non-dimensional frequency of the vortex breakdown was evaluated. The Strouhal number obtained on the basis of numerical simulation, S = 0.21, was equal to literature data [2].

(a)



(b)

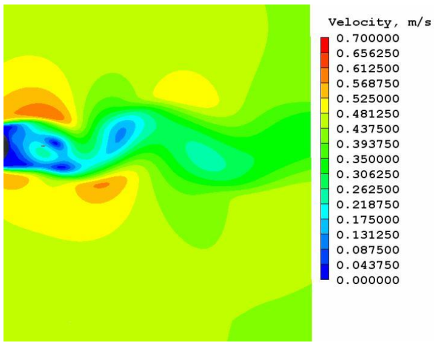


Fig. 2. Whole velocity field snapshots obtained by the experimental data analyze (a) and the numerical simulation (b).

3. Conclusions

The new course, combining the basis of numerical and experimental research, was presented to the Master Degree students in Saint-Petersburg State Polytechnical University.

The numerical simulation of classical aerodynamic problems was performed by PHOENICS 2008. It proved to be comfortable software for the students. Its interface was acknowledged to be intuitively comprehensible, even by students who did not specialize in numerical simulation.

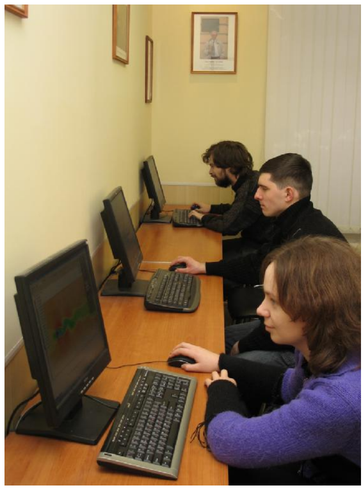


Fig. 3. PHOENICS lesson in the CFD class.

Experimental investigations were fulfilled using the POLIS complex based on the PIV technique. The author would like to express thanks to experimental team members: Prof. S. Z Sapozhnikov, Prof. V. U. Mitiakov, Dr. A. V. Mitiakov & MSc S. A. Mozhayskiy.

4. References

1. POLIS description / Institute of Thermophysics SB RAS http://www.itp.nsc.ru/piv/piv.htm
2. Schlichting G. Boundary layer theory. M.: Nauka, 1974 – 712 c.
3. P.K. Chang. Separation of Flow. – M.: Mir, 1972.– 300 c.

Department of Thermodynamics and Heat Transfer,
Power Engineering Faculty, SPbSPU
Polytechnicheskaya 29, Saint-Petersburg, Russia
Phone/Fax: +7 812/552-7773
e-mail: tot@pef.spbstu.ru, http://www.spbstu.tot.ru

2.2 F1-VWT Report by Tyler Gagat (previously of East Cobb Middle School, Georgia, USA)

As a fourteen year old I would never have expected to be using CFD software to design miniature Formula One cars, and would expect even less to have been presenting my finding to a panel of judges at the International Autosport show in Birmingham England.

Five years later my knowledge of CFD software has taken me around the world multiple times, introduced me to major F1 leaders, provided me with an internship at an aerospace firm, and given me the opportunity to share this technology with hundreds of other students.

Since that first World Championship at the Autosport show our teams have competed in three other World Championships; in Australia, Malaysia and London, and have even held F1 In Schools races at two United States Grand Prix. One such Grand Prix led to a race between our team and a small team of Ferrari engineers headed by Ross Brawn himself.

Flying Cougars F1 in Schools team, racing Ross Brawn at 2006 US Grand Prix

My role on the team was that of CFD engineer, I did aerodynamic analysis on every car developed by all of our teams helping them to better understand which designs worked and which ones didn't. Our teams did almost all of our aerodynamic testing using CFD software from CHAM, testing dozens of designs and dozens more variations of each design to reach a final product.

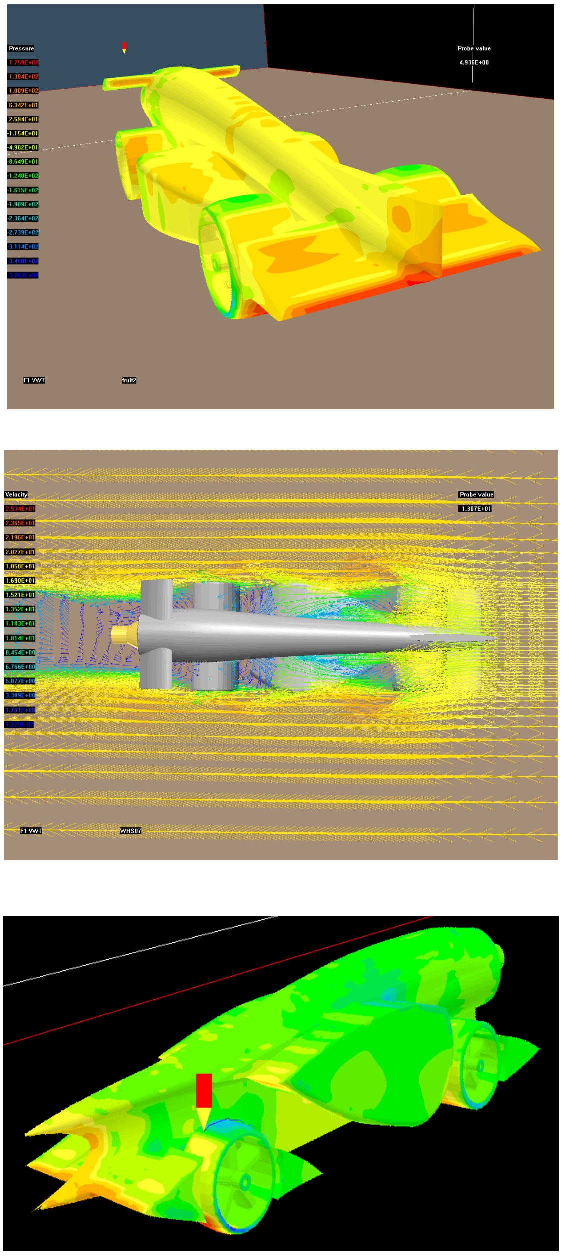
But we didn't just do this for our teams, our school had the resources and the knowledge to help multiple other teams from surrounding schools, and in addition to our teams we also trained hundreds of students from other schools how to design, analyze and manufacture F1 in Schools cars.

Team Hybrid USA and Scotland Team at the F1 in Schools World Championships in Kuala Lumpur, Malaysia 2008

Over those five years working with CFD in F1 in Schools my interest in the field of aerodynamics continuously grew. It influenced me to take more engineering oriented classes in high school, to do research in the field of aeronautics and to apply to jobs and universities with an engineering focus. My senior year of high school I secured an internship at SpaceWorks Engineering Inc. a company that does research work for NASA and companies like Lockheed Martin. At this job, I was able easily to enter a professional environment and actively work on their CFD software writing a tutorial for the company on their new CFD and GUI software, using my previous experience with CHAM software.

My final steps of high school involved applying to colleges. Because of my experience, both in F1 in Schools and my internship, I was accepted to every school I applied to, including multiple top colleges in the United States as well as City University of London's Automotive & Motorsport Engineering Program.

Three Images of our 2007 Cars in PHOENICS



Supplied by Fred Stillwell
East Cobb Middle School, Georgia BEST
F1 in Schools USA In Country Coordinator

3) Technical News & User Applications from Agents

3.1 ChemTech Technical Report: from IMI Brazil by Ardson dos Santos Vianna Junior

1. Introduction

The Military Institute of Engineering (IME) was established in 1792, by the King of Portugal. The history of IME is intertwined with the history of military education and teaching engineering in Brazil. The Institute has its origins in the Royal Academy of Artillery, Fortification and Design, established in 1792 to serve the interests of the Portuguese Crown in Brazil. It was the first School of Engineering established in the Americas and the third in the world. From the year 1964, it started to admit civilian students in all departments. At present, the institute has civilian and military students.

The institute was originally governed by the Brazilian Army. There are six departments covering Civil, Electronic, Mechanical, Chemical, and Cartographic Engineering, and Computer Science. The Chemical Engineering Department is responsible for graduate (Chemistry) and undergraduate (Chemical Engineering) courses. Our research focuses upon: Organic Syntheses, Physical Chemistry and Chemical Technology.

2. People

The CFD Group is coordinated by Prof. Dr. Ardson dos Santos Vianna Jr. and is composed of:

Graduate students:

1. Evandro de Souza Nogueira, D.Sc., since 2007;
2. Genizia Islabão de Islabão, D.Sc., since 2007;
3. Jorge Luiz da Silva Porto de Oliveira, D.Sc., since 2008;
4. Giancarlo Cantaluppi Silvestre de Freitas, D.Sc., since 2008
5. Ilmar Victor , D.Sc., since 2009;
6. Isis Campos Prado, M.Sc., since 2008.

Undergraduate students:

1. Raphael Pereira Scudino Borges

3. Collaborators

The CFD group has worked with CTEx, Army Technology Centre, and Professors from other Universities:

a) José Carlos Pinto PEQ/COPPE-UFRJ
b) Fernando Cunha Peixoto UFF

4. Before 2009

The IME CFD Group has been using PHOENICS at graduate and undergraduate levels, since 2004. The M.Sc. Theses are:

1. Modelagem Matemática de Sistemas Eletroquímicos Utilizando Ferramentas de Fluidodinâmica Computacional, por Giancarlo Cantaluppi Silvestri de Freitas em 2005.
2. Modelagem da Dispersão de Agentes Químicos Através de Técnicas de Fluidodinâmica Computacional por Leandro Radusweski Quintal em 2006.

Senior undergraduate projects are:

1. CFD Modelling of Continuous Tank, by Rafael Duarte Barros e Carolina Potasso Braga, 2004.
2. CFD Modelling of Gas Dispersion, by Jório Gottardo Jadjiski, 2005.
3. CFD Modelling of Continuous Clarifier, by Reuel Lopes Paula and Edilson Gama dos Santos, 2007.

Junior undergraduate projects are:

1. Turbulence Models In A Practical Case: The Refrigeration of A Classroom by Caio Veiga Penna Delgaudio, 2008.
2. CFD Modelling of Continuous Clarifier by Caio Veiga Penna Delgaudio, 2009.
3. Trajectory Of Particles Within A Continuous Thickener, by Vitor Freitas Lima Burjack, 2009.

5. From 2009 to now

These D.Sc Theses are still in progress:

1. CFD Modelling of Tank Polymerization Reactor, by Evandro Nogueira.
2. CFD Modelling of Tubular Polymerization Reactor, by Genizia Islabão de Islabão.
3. Gas Dispersion Subjected to a Moving Train, by Jorge Luiz da Silva Porto de Oliveira.
4. Gas Agent Release Using CFD, by Giancarlo Cantaluppi Silvestri de Freitas.
5. Biological Agent Release Using CFD, por Ilmar Victor Marinho Barbosa.

6. Group Results 2009

6.1 Study of a Continuous Thickener's Geometry

Abstract: This paper uses the techniques of computational fluid dynamics to evaluate a continuous thickener. Velocity fields were obtained for various configurations, varying the parameter H / R (height of the thickener on its radius) and height of the break-flow (Hqf), analyzing the synergy of these variations.

A case has also been evaluated considering the drop in the bottom of the break-flow. The analysis conducted to a discussion on details of equipment design that is not normally considered in a conventional project. For the simulations, the software PHOENICS 2008 was used as it proved to be effective for the desired analysis.

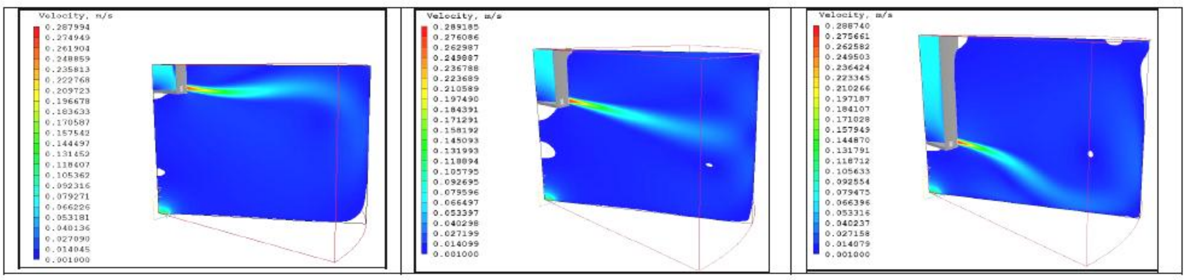


Fig. 1: Velocity profiles to different heights of the break flow.

6.2 Simulation Of Multiphase Liquid-Liquid Suspension Systems Using Computational Fluid Dynamics

Abstract: Using CFD tools, this work evaluates the spatial distribution of the rate of turbulent dissipation energy (ε) and of the normalized constant of breakage in a liquid-liquid suspension system maintained in a tank agitated by a four-bladed impeller. The qualitative effect of the spatial distribution of ε on the breakage constant, usually defined as a function of the average ε value, is also analyzed. The use of the compartmentalization model (which assumes that the tank can be represented in terms of two or more isotropic regions or zones) is also evaluated. The obtained results indicate that particle breakage occurs in a tiny region around the impeller and close to the wall of the vase. Therefore, the rate of breakage can be regarded as a boundary condition for the particle population balance and it does not seem reasonable to assume that the breakage takes place in an isotropic medium. On the other hand, the coalescence rate can be defined in terms of the characteristic average ε value in the tank. Simulations were performed with the commercial package PHOENICS 2008, developed by CHAM Limited. It is assumed that the multiphase system follows the two-fluids model.

Turbulence was modelled in accordance with the RANS methodology and the model k- ε RNG model.

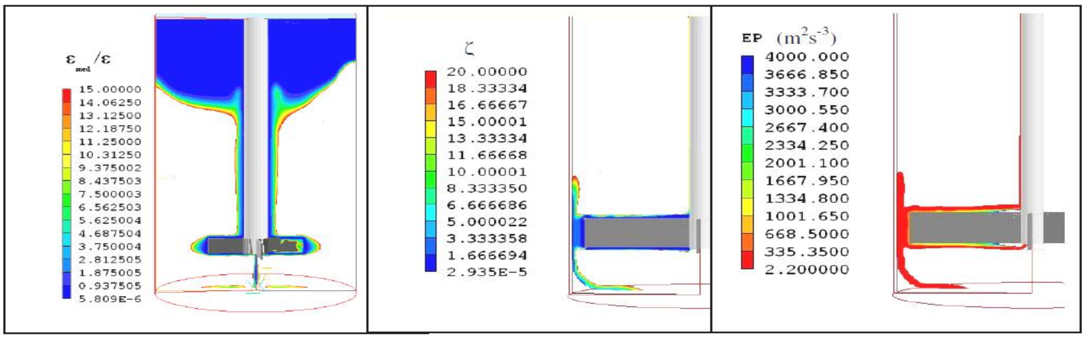


Fig. 2: the rate of turbulent dissipation energy.

6.3 Comparative Study of Tubular Reactors with Different Geometries Using CFD

Abstract. The simulation of polymerization in a tubular reactor is very important for industrial applications. Two approaches (linear and helical geometry) were compared for a tubular reactor, where they obtained the velocity profiles along the reactor. The geometry directly influences the profiles of velocity of the fluid. In the case of helical geometry, was observed that the fluid that is in contact with the inside of the curve shows a flattening of the current lines, which changes the displacement of fluid through the reactor and consequently the degree of mixing. This fact was not observed in the case of linear geometry, since the streamlines presented symmetry within the tubular reactor. In addition, patent applications filed in different countries were mapped, which identified the technological development over the years, focusing on the use of this simulation tool (Computational Fluid Dynamics - CFD), both in the polymer and in other areas of knowledge.

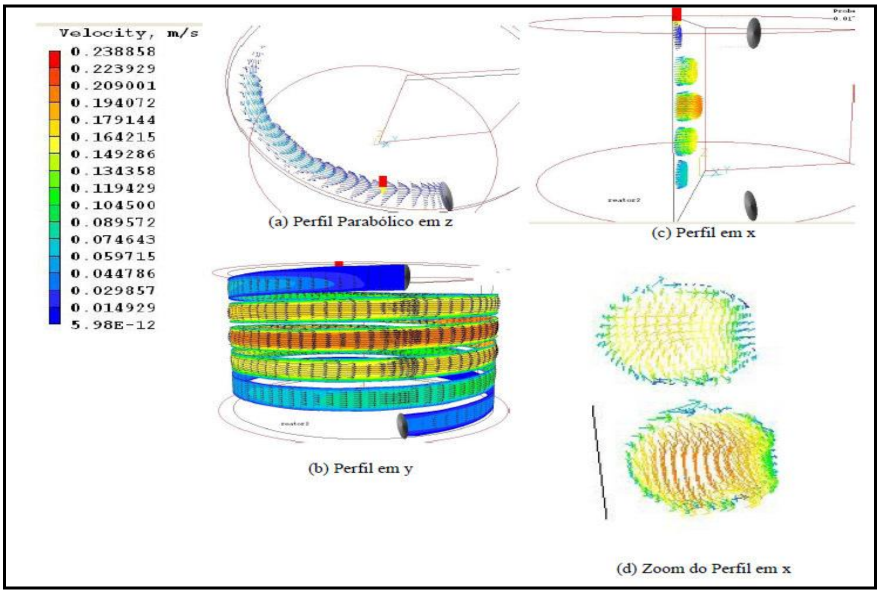


Fig 3: Velocity Profiles to Helical Reactor

6.4 Gas Dispersion Subjected to a Moving Train

Abstract. The answer to chemical and biological attacks relies on the ability to monitor and detect the presence of an agent. Dispersion modelling and simulation of contamination by warfare agents through the calculation of flow fields and concentration distributions can be achieved using computational fluid dynamics (CFD) tools. The aim of this study is to evaluate the dispersion of chlorine gas in a Brazilian subway station subjected to a moving train. The movement induces additional kinetic energy that can be used as an alternative extraction system. The visualization of the velocity fields and toxic-gas-concentration profiles showed that the action of the moving train is very quickly and practically one-dimensional. The plume was modified by the velocity-

vector field, which is generated by the moving train. These effects were more pronounced upstream and downstream than upside and downside. Therefore, CFD can be used to propose a contingency plan for potential accident scenarios.

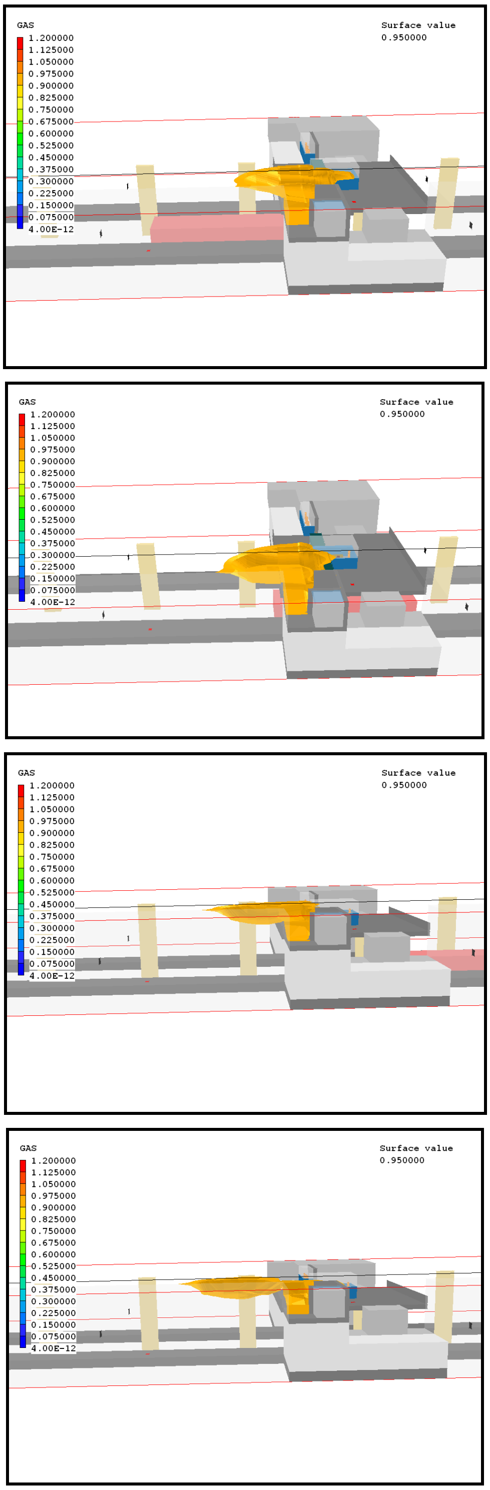


Fig. 4: Iso-concentration surfaces of the contaminant at the time step corresponding to dispersion phenomena when train passes through the mezzanine: (a) 16.8 s; (b) 17.76 s; (c) 18.72 s; (d) 21.6 s;

7. Submitted articles

1. Latin American Applied Research, “Turbulence Models In A Practical Case: The Refrigeration of A Classroom”, 2009.
2. Applied Mathematical Modelling, “Gas Agent Release Simulation Using CFD”, 2009.

Military Institute of Engineering, Chemical Engineering Department, ChemTech - Brazil

3.2 Chemtech Technical Report: UFU Brazil

IMPLEMENTATION OF TURBULENCE MODELS FOR HYBRID AND URANS METHODS IN COMMERCIAL SOFTWARE PHOENICS by

COORDINATOR: Aristeu da Silveira Neto
EXECUTORS: Diego Alves de Moro Martins, Elie Luis Martínez Padilla, João Marcelo Vedovoto

1. INTRODUCTION

Many applications in engineering involve turbulent flow with high Reynolds numbers over complex geometry. To improve analysis of these applications it is necessary to use turbulence models.

In this present contribution two types of turbulence models were used: URANS (Unsteady Reynolds-Averaged Navier-Stokes) and Hybrid (RANS-LES) in the PHOENICS software.

The URANS models implemented are: SST (Shear Stress Transport), Menter (2003b), and SA Spalart-Allmaras (1997), turbulence models. The hybrid models implemented are: SST-DES (Datched Eddy Simulation) and SA-DES turbulence models.

2. IMPLEMENTATION

Turbulence models in PHOENICS were implemented via Inform in group 13 of the Q1 file, so turbulence viscosity is returned after the calculus.

Routines were developed specifically for two CFD (Computational Fluid Dynamics) cases, driven cavity and backward-facing step. The temporal regime is transient, and the domain is three-dimensional for both cases. Temporal terms were solved by an explicit first-order method. Spatial terms were discreted explicitly by the up-wind first-order method. To put in contour conditions "Patches" were created in the wall regions, as shown in Figure 1.

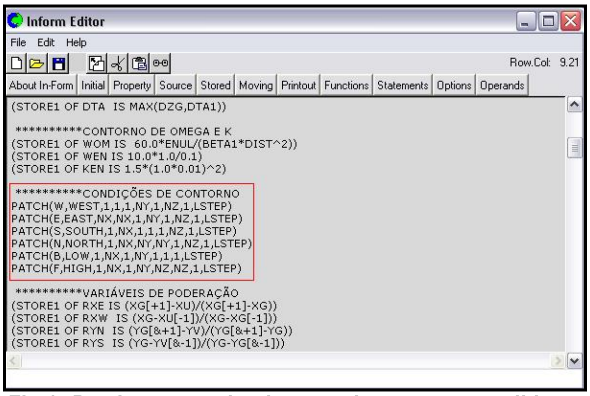


Fig 1. *Patches* created to impose the contour conditions.

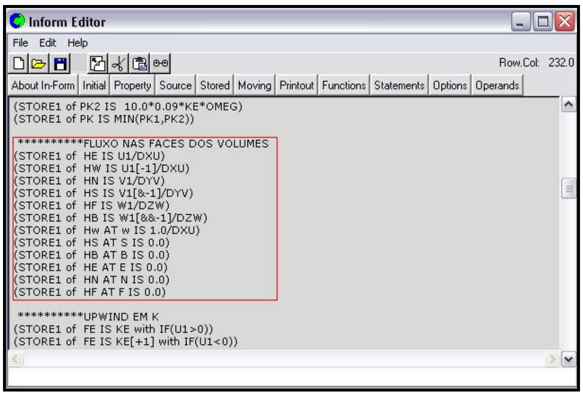


Fig 2. Flux calculus

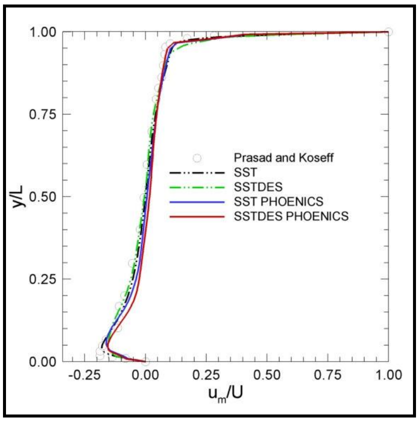
It was necessary to impose flux values in the contour domain because, when these fluxes are calculated in the advective term, the solution diverges. Figure 2 shows this imposition.

To generalize the implemented cases, it is necessary to write the contour conditions of the turbulence models implemented into the base code of PHOENICS.

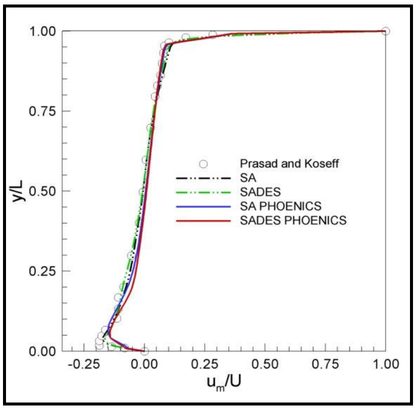
3. RESULTS

a. Driven Cavity

To validate what was done, results of the models implemented in the PHOENICS were compared with experimental results (Prasad e Kossef, 1989), and also with results of our own codes. Figures 3 and 4 show the velocity profiles on a middle plan, comparing results obtained. The Reynolds number of this case is 10000.

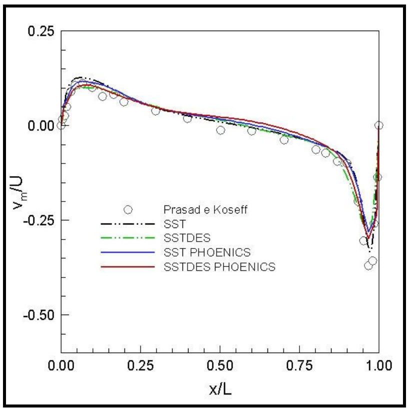


(a)

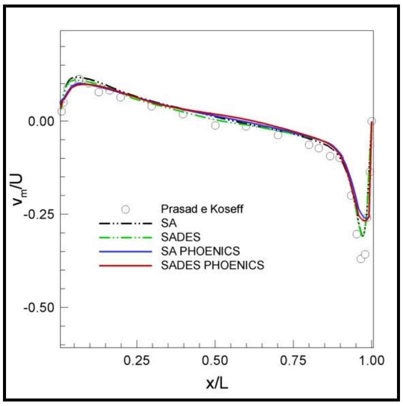


(b)

Fig 3. Velocity profile in the x direction in a middle plane.



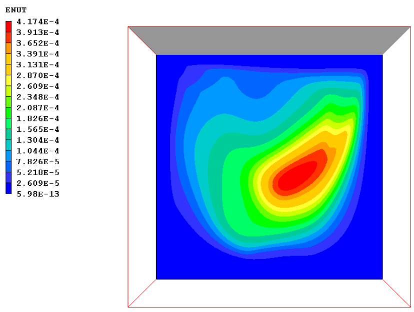
(a)



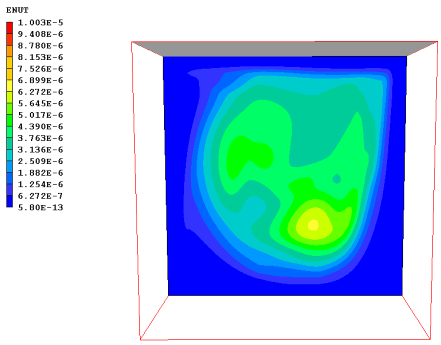
(b)

Fig 4. Velocity profile in the y direction in a middle plane.

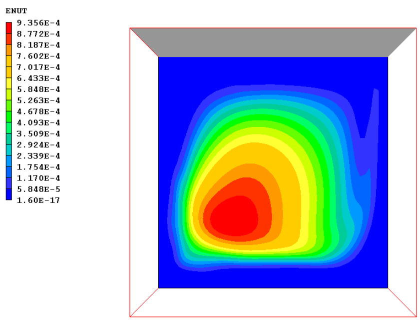
Agreement was good between the turbulence models implemented and experimental results in respect of the driven cavity results. Figure 5 shows the turbulent viscosity contour in a middle plan.



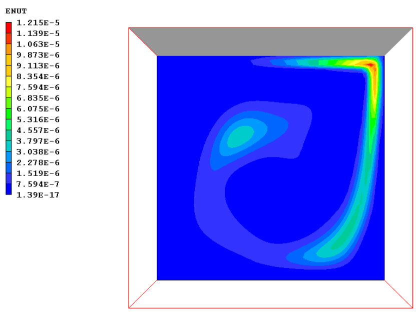
(a)



(b)



(c)



(d)

Fig 5. Turbulent viscosity contour in a middle plan for the turbulence models implemented: SST (a), SST-DES (b), SA (c) and SA-DES (d).

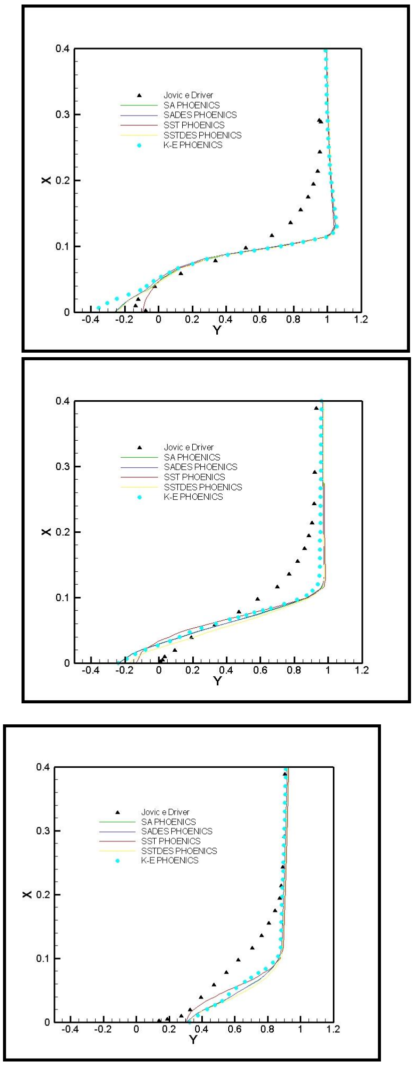
b. Backward-facing Step

To validate the turbulence models implemented in this case, a comparison was made between models implemented, models existing in PHOENICS and experimental results (Jovic & Driver, 1994). The backward-facing step parameters are in Table 1.

Table 1. Simulation parameters.

Re	Domain	Mesh	Time Step
5000	2.30x0.6x0.4	80x40x20	1E-02

Figures 6a, 6b and 6c show respectively the x-velocity profile in the planes x=4, x=6 and x=10.



(a)

(b)

(c)

Fig 6. Velocity profile in a middle plan over y, in x = 4 (6.a), x = 6 (6.b) and x = 10 (6.c).

The results of the turbulence models implemented are consistent compared with the models in PHOENICS but, when compared to experimental results, a difference was observed due to the fact that it was not possible to simulate the correct parameters of the experiment.

Figures 7, 8, 9 and 10 show respectively the turbulent viscosity contour of the SST, SST-DES, SA and SA-DES turbulence models.

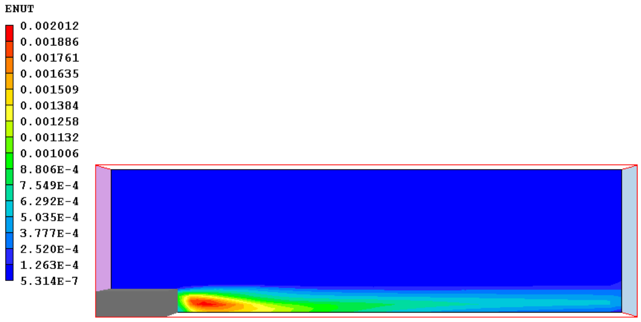


Fig 7. Turbulent viscosity contour of the SST turbulence model.

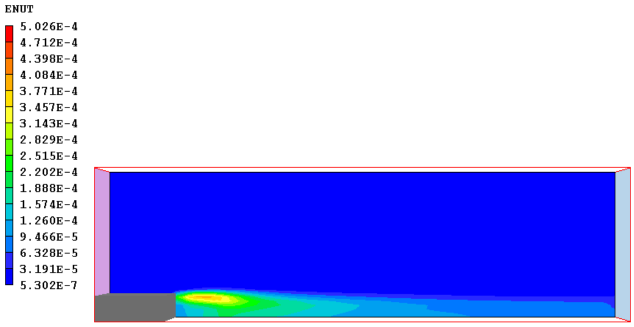


Fig 8. Turbulent viscosity contour of the SST-DES turbulence model.

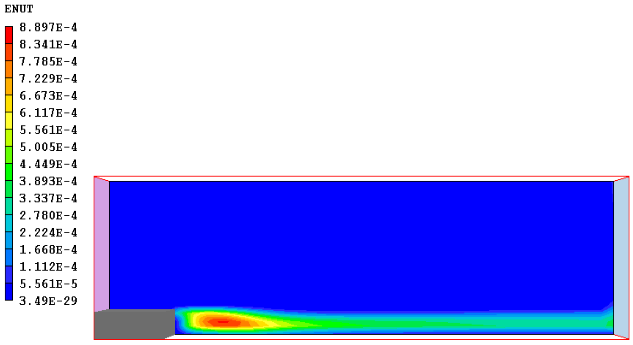


Fig 9. Turbulent viscosity contour of the SA turbulence model.

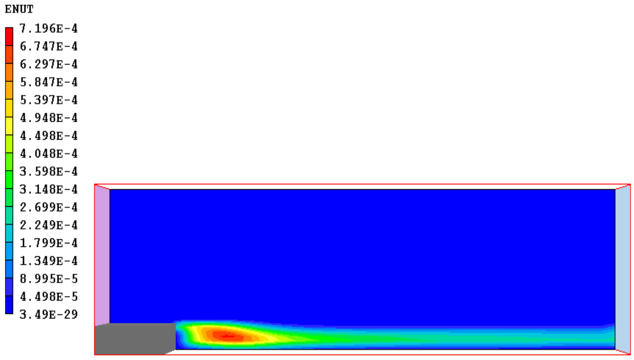


Fig 10. Turbulent viscosity contour of the SA-DES turbulence model.

4. CONCLUSION

For this paper four turbulence models from PHOENICS were implemented and validated namely SST, SST-DES, SA and SA-DES. The validations were made through two CFD cases, the driven cavity and the backward-facing step.

Results obtained for driven cavity are satisfactory compared to experimental results. In the backward-facing step there was a difference between the experimental results and the PHOENICS results, due to the fact that was not possible to set the experimental parameters in PHOENICS. Conversely, the results of turbulence models implemented are consistent compared with the models existing in PHOENICS.

The implementations are specific to the cases analyzed. To generalize the results it would be necessary amend the base code of the PHOENICS software.

5. REFERENCES

1) Jovic, S., Driver, D., 1994, "Backward-Facing Step Measurements at Low Reynolds Number, Reh=5000", NASA Technical Memorandum 108807.

2) Menter, F.R., Kuntz M. and Durand, L., 2003b, "Adaptation of Eddy-Viscosity Turbulence Models to Unsteady Separated Flows Behind Vehicles", The Aerodynamics of Heavy Vehicles, United Engineering Foundation.

3) Prasad, A.K. and Koseff, J.R., 1989, "Reynolds Number and End-Wall Effects on a Lid-Driven Cavity Flow", Phys. Fluids A, 1, pp. 208-218.

4) Spalart, P., S. Jou, W.-H., Strelets, M. & Allmaras, S., 1997, "Comments on the Feasibility of LES for Wings & on a Hybrid RAN/LES approach", Advances in LES/DNS, First Int Conf on DNS/LES, Louisiana Tech University: Greyden Press.

Fluid Mechanical Laboratory, Federal University of Uberlândia

3.3 Coolplug Technical Report

Natural ventilation of an indoor athletic training hall (Hamburg, Germany).

Photo:Jürgen Schmidt, Hamburg

1) Goal

The goal was to provide a natural ventilation solution for this fully glazed hall, despite the significant solar heat loads. Mechanical ventilation was not allowed. For heating underfloor heating was planned. The air exhaust is through light domes in the roof.

2) Procedure

1. Analysis of the thermal behaviour of the hall with a thermal simulation program (EES, like TAS from EDSL).
2. 3D-Modelling of the hall including surrounding buildings in PHOENICS.
3. Determining of the pressure conditions at the hall for various wind velocities and directions. From these PHOENICS calculations it was concluded that no satisfactory solution could be obtained by mounting hopper windows and the light domes. To stabilize the ventilation wind catchers and motor-controlled glass louvers were chosen.
4. Flow simulation of wind catcher and glass louver to determine the pressure loss coefficients.
5. Simulation of winter- summer- spring/autumn operation (flow and temperature) with PHOENICS.

3) Conclusion

The PHOENICS simulations showed an excellent thermal behaviour of the indoor athletic training hall on the condition that control of the glass louvers, hopper windows and the valves in the wind catchers provides a uniform and wind independent ventilation of the hall.

Another result of the simulations was that the temperature in the middle of the hall was too high in winter due to the underfloor heating. This can be avoided by a different distribution of the tubes of the underfloor heating.

Photo:Jürgen Schmidt, Hamburg

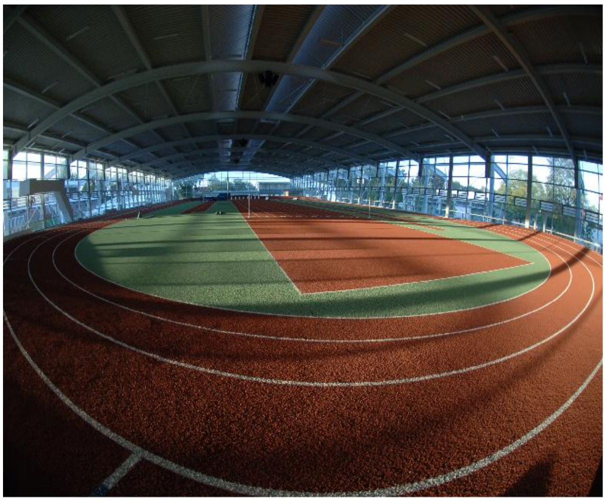
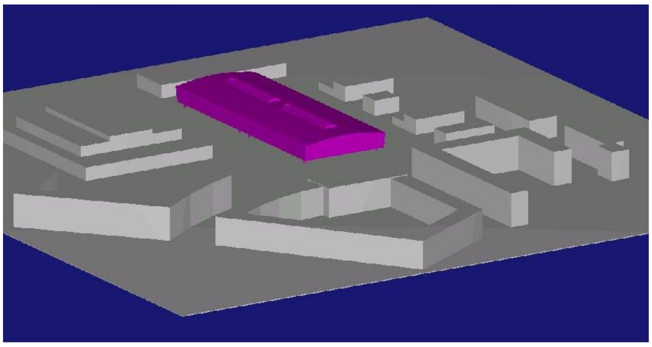
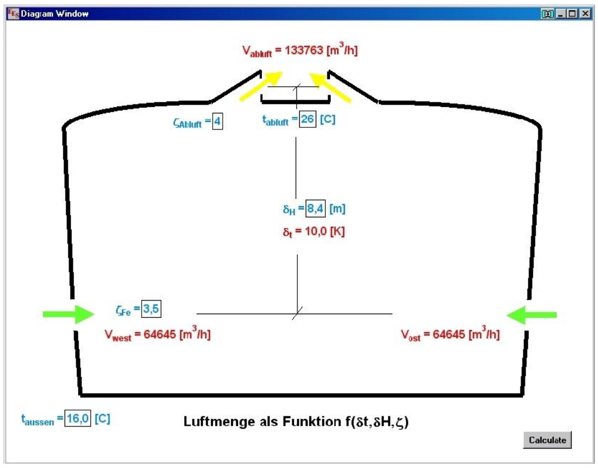


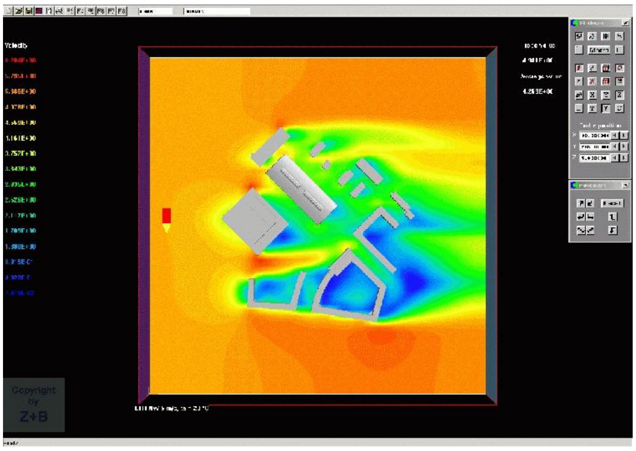
Photo:Michael Zapf, Hamburg



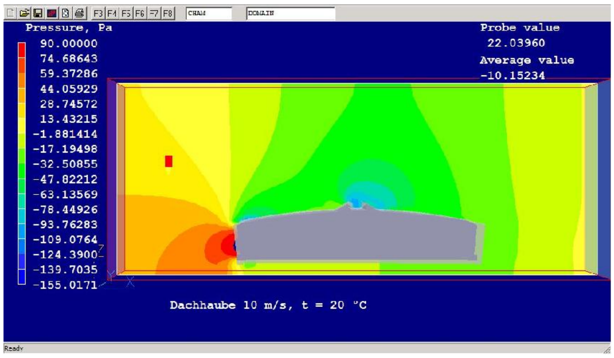
Modelling of the athletic training hall and the surroundings



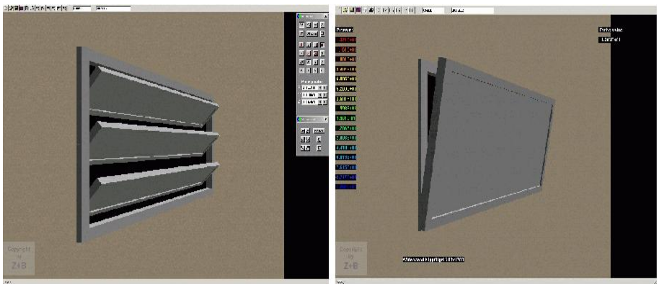
Sketch of flows through the hall



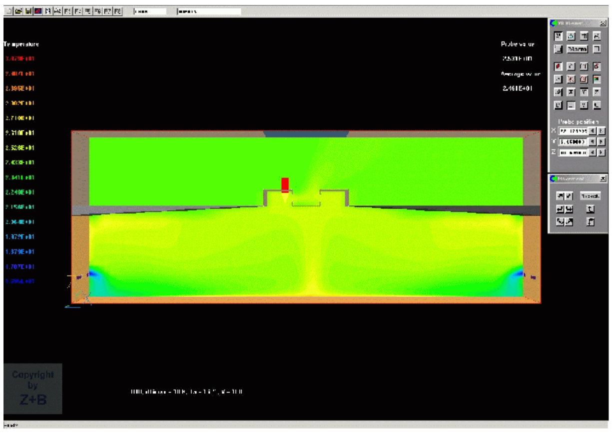
Air velocities for NW 5 m/s wind, 20 degrees C



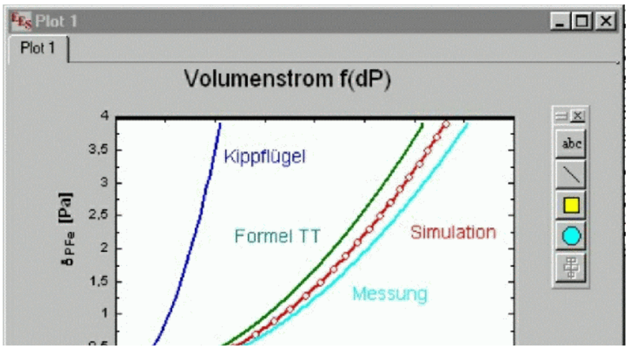
Example of pressure distribution around the hall. Wind coming from left towards the side wall.



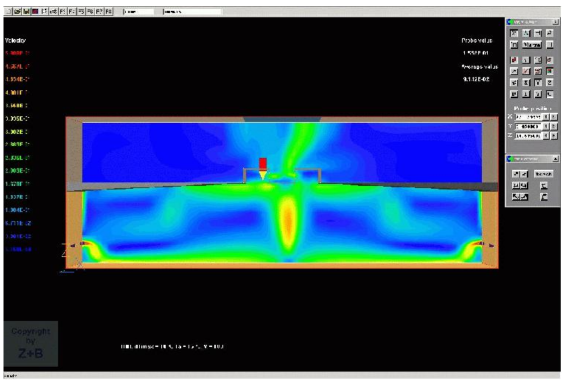
Modelling of glass louver and hopper window to determine pressure loss coefficients



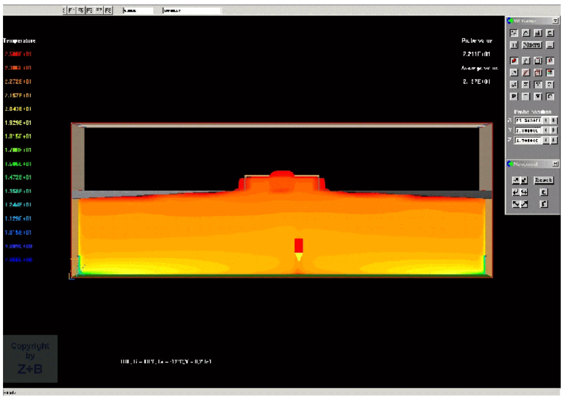
Temperature distribution in spring



Comparison of simulation and measurement (light blue. Messung) for a glass louver



Velocities in spring



Afbeelding 1: Temperatures in winter.

3.4 Coolplug Technical Report

Fashion Museum of Ludwigsburg Palace, (Ludwigsburg, Germany).

Ludwigsburg Palace

In the Fashion Museum of Ludwigsburg Palace a fashion show featuring clothing from the 18th century to the today is presented with the latest in museum technology. The exhibition pieces are in air-conditioned glass cabinets. The exhibits are not allowed to move due to air flow. The air flow must be uniform and completely draught-free in the region of the exhibits. The required relative humidity is 50%.

1) Goal:

The goal was to develop a draught-free air inlet for every separate glass cabinet.

2) Procedure:

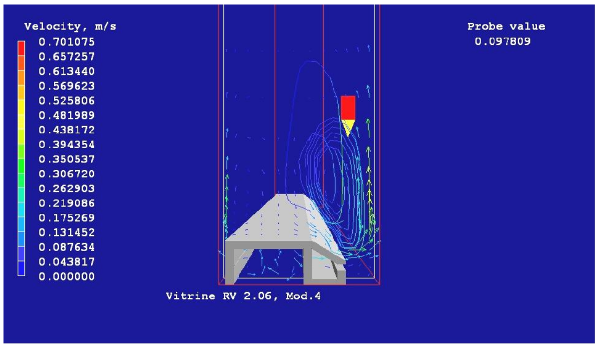
PHOENICS was used to perform flow simulations to prevent, absolutely, mechanical loads on the very old exhibits.

3) Conclusion:

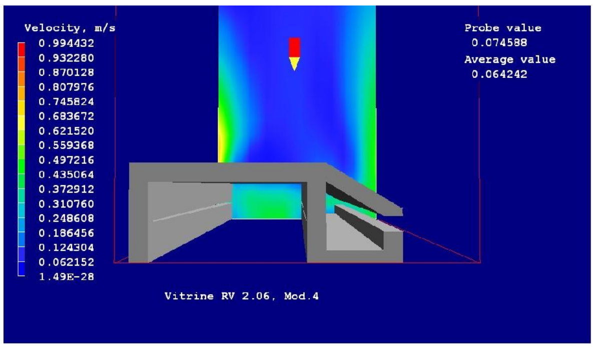
By using PHOENICS the air inlets could be designed completely draught-free and independent of the geometry of the glass cabinets. In nearly all 50 cabinets the demand was fulfilled that no load had to be exposed to the exhibits by air movement. Costs for expensive tests could be avoided and a modular solution for all cabinets could be found.

Large glass cabinet.

Glass cabinets at the stairs to the first floor.



Streamlines in a raised floor of a long cabinet



Flow velocities in a raised floor of a long cabinet

4) News and Events 2010

4.1 Courses and Meetings

Drs John Ludwig & David Glynn delivered a PHOENICS/FLAIR training course in February at CHAM London. Training courses comprise a three-day programme with extra workshop time. Participants share experiences and present their problems to CHAM staff, so that solutions can be reached and working methods explored. The next CHAM course is in Wimbledon, April 13 – 15 2010. For further information, or to book a place, please contact sales@cham.co.uk.

4.2 News from Down Under by Murray Mason

ACADS-BSG, CHAM's agents in Australia, will be exhibiting PHOENICS along with the other building services programs that it distributes at the 2010 arbs Trade Exhibition, Australia's major national and international air-conditioning, Refrigeration and Building Services trade exhibition. Show casing a diverse range of goods and services from the HVAC & R and building services and associated industries, arbs is a three day mega event that brings together all segments of the industry every two years. This year the exhibition is to be held in the Sydney Convention & Exhibition Centre from April 12-14 and there will be over 200 exhibitions

This Newsletter is a Platform for You to tell Us, and Others, what You are doing with PHOENICS

Commercial PHOENICS Users are invited to email articles for inclusion on the CHAM website and in the next Newsletter to cik@cham.co.uk.

Academic PHOENICS Users are also invited to contribute and are reminded that CHAM looks forward to receiving, from each of them when their Academic licences are renewed, a report of work carried out with PHOENICS- which can be published without copyright issues. Reports can be sent directly to mjl@cham.co.uk or to the relevant CHAM Agent.

Please Send us Your Contributions. Thank You

4.3 Current and forthcoming Events 2010

Feb 18	PHOENICS CFD Trial Seminar, CHAM Japan, Tokyo (unless otherwise specified)
Feb 19	PHOENICS Basic Course, CHAM Japan
Feb 24	MOFOR Course, CHAM Japan
Mar 03	Free Surface Flows Course, CHAM Japan
Mar 11	Gentra Course, CHAM Japan
Mar 12	IPSA & Two Phase Flow Courses, CHAM Japan
Mar 18	PHOENICS CFD Trial Seminar, CHAM Japan
Mar 26	PHOENICS User Seminar, CHAM Japan, Kansai PHOENICS Hands-On Seminar, CHAM Japan, Osaka
Apr 09	CVD Course, CHAM Japan
Apr 13 -15	PHOENICS/FLAIR Course, CHAM London
Apr 15	PHOENICS CFD Trial Seminar, CHAM Japan
Apr 16	PHOENICS Basic Course, CHAM Japan
Apr 26 -30	Franklin Medal Award Ceremony, Philadelphia, Pennsylvania
Jun 09 -11	8th International ERCOFTAC Symposium on Engineering Turbulence Modelling & Measurements, Marseille
June	Shanghai User Meeting & Shanghai Expo. Training Courses Beijing, Shanghai, Xi'an, Shenshen. Dates to be announced
Jun 23 -25	DMS, 21st Design Engineering & Manufacturing Solutions Expo, Tokyo

4.4 CHAM Staff News

Professor Spalding is busy outside, as well as within, CHAM. He has been appointed to the Awards Panel of the Global Energy Committee for a further 5 year period and has agreed to edit Section 1 of the Heat Exchanger Data Handbook (HEDH) published by Begell House. He will lecture on *Benjamin Franklin and CFD* at the Franklin Awards Ceremony in April and on *PPT and PPB fields, facilitating the comparison of experimental and DNS-, LES-, PANS-, PDF-transport or MFM-model representations of turbulence* at the 8th International ERCOFTAC Symposium in Marseille, June 9-11 2010.