PHOENICS News

Spring 2011

1) Editorial

New faces in the CHAM team...

Paul Emmerson, Rama Devi Pathakota, Cinzia Taccoli and Tomasz Stelmach

Three new members of staff joined the Consultancy Team at CHAM in Wimbledon in January. They are:

Rama Devi Pathakota
Education

- Bachelor of Engineering (BE) in Mechanical Engineering from Osmania University, India. (2005-2009) Aggregate: 77.3%
- MSc in Computational Fluid Dynamics from Cranfield University, UK (2009-2010) Aggregate: 74%

Expertise

- Numerical modelling of industrial heat exchangers.
- External aerodynamic flows.
- Design and optimization of high-lift devices.
- Numerical methods and PDEs.

Cinzia Taccoli is completing her PhD entitled "Experimental and Computational Analysis of Purge Systems for Radiation Pyrometers" at Cranfield University. The research involves CFD and FEA analyses integrated by PIV experiments.

After completing her BSc in Aerospace Engineering and her MSc in Space Engineering at the University of Rome "La Sapienza" she collaborated with the University on the design of an integrated system for satellite energy storage and altitude control. During this period she was also a member of the system-engineering team of the European Student Moon Orbiter, an educational project run by the ESA Education Office supported by SSETI.

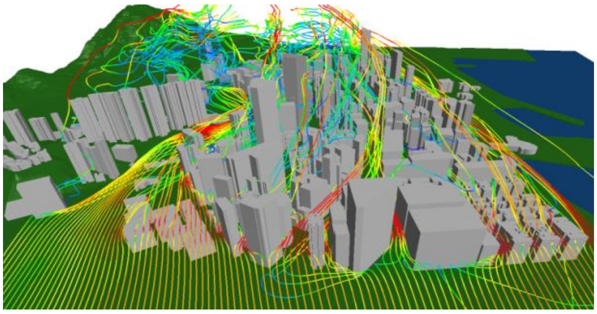
Tomasz Stelmach obtained an MSc in Power Engineering at The Silesian University of Technology in Gliwice, Poland in 2008. He took part in the students' exchange programme "Erasmus" and spent his last semester at the Centre for CFD in the University of Leeds where he finished his Master's thesis. In January 2009 he was offered a position as a Research Assistant at the University of Leeds. His main responsibilities included development and application of computer models of power generation plants with carbon capture installations. He was also appointed a CFD consultant in a coal and biomass combustion project. Tomasz is currently working on his PhD.

2) News and Events

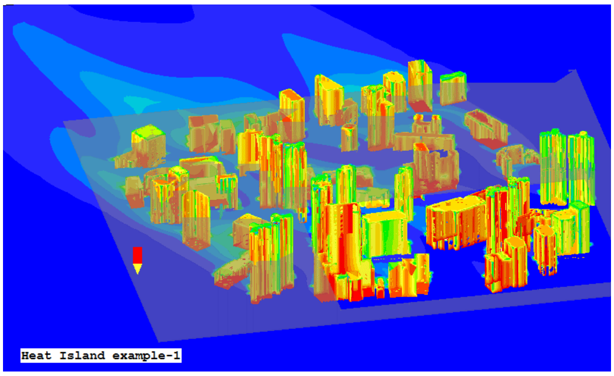
2.1 CHAM News

2.1.1 High Performance Cluster News

The Figure below shows hilly terrain behind Times Square – exaggerated height – streamlines @ 20m coloured by velocity.



The Hong Kong Planning Department (HK PlanD) has purchased another multiple-copy parallel-processing licence of PHOENICS/FLAIR for installation on its second multi-core High Performance Cluster. This follows a similar installation of PHOENICS at PlanD in late-2009, where the software is utilised by the Urban Design & Landscape Section for modelling large-scale urban wind studies.



This new procurement, by the Information Systems & Land Supply Section, will be used as a tool for urban heat island (UHI) research. New features have already been added to PHOENICS/FLAIR to facilitate this, with more to be added as part of a collaborative programme, to be implemented as HK PlanD's longer-term UHI research requirements become apparent.

CHAM agent, Lasertec Srl has signed a joint-collaboration agreement with the Politecnico di Milano. This provides a centre of excellence and education for PHOENICS located at the university and more wide-spread availability of PHOENICS installed upon the huge CILEA High Performance Cluster.

The expansion of PHOENICS into China continues to gain momentum. Having reported excellent results for 2010, CHAM's agent Shanghai Feiyi recently confirmed that its latest customer, the National Centre for Quality Supervision & Inspection of Traffic Safety Facilities, will be installing PHOENICS on its 128-node High Performance Cluster. This new sale represents the largest implementation of parallel-PHOENICS in China.

And what of CHAM itself?

In response to the increasing demand for the purchase and support of parallel-PHOENICS installed on both Windows-based and Linux-based HPC's, we have ordered a more modest 32-core dual-boot HPC configuration.

2.1.2 PHOENICS Training Courses

The next PHOENICS/FLAIR Training Courses will be held at CHAM in Wimbledon on May 17-19 2011 and July 19-21 2011. These are CHAM's regular courses and comprise a three-day introduction to PHOENICS although attendees may come for only part of that time. Please contact Peter Spalding on PHOENICS@cham.co.uk or email e-mail sales@cham.co.uk to register for the next training course or to obtain further information. Courses can also be run at customer sites by prior arrangement

2.1.3 CHAM Website

CHAM is working on a new website which, it is hoped, will go live on May 1. We want to ensure that we have the most up-to-date links to the PHOENICS element of all our Agents sites so please would each of you email these links to wbscham@gmail.com. Also if you would like us to include a brief description of your PHOENICS operation on our website please also send that, by email, to the above address.

2.1.4 CHAM Sponsorship

CHAM Staff wearing red to support British Heart Day.

2.2 Branch and Agent News

2.2.1 CHAM Japan

Our main concern upon learning about the earthquake, and subsequent tsunami, in Japan on March 11 was to make contact with CHAM-Japan to see that our staff was safe.

Fortunately they were and, as you can see from the message behind the photo of some of the staff in the Tokyo office above, they appreciated the interest and expressions of concern. Mr Zuwei Kong, the Manager at CHAM Japan, sent a message to say that:

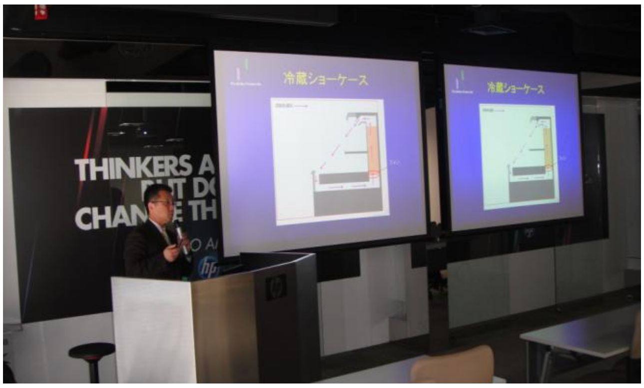
"After the earthquake on 11, March, 2:46 PM, our office closed at about 5pm and all of our staff walked home because the trains were stopped. One staff member took 10 hours to walk to home. Our office was not damaged by this earthquake although some books and files fell off the desks to the floor.

Monday of the following week, our office was closed for one day and opened from Tuesday. For that week, although public transport was not normal and the nuclear plant accident still remained an issue, our office worked normally."

A PHOENICS User Conference was held on January 28 2011 at the HP Seminar Centre in Tokyo. Some 40 Users attended the Conference at which CHAM-J presented the new features of PHOENICS 2010. Two photographs are shown above and below.

Five users presented their PHOENICS research and the topics of their presentations are listed below:

1) Flow Simulation of the Air Curtain in Supermarket Refrigerator Showcase, Ryokoku University
2) Flow Simulation Application for Cutting Thick Steel with Lasers, CHIBA University
3) Simulation for Marangoni Convection around a Boiling Water Bubble, Shibaura Institute of Technology
4) Heat Flow Simulation of a Thermal-Storage Medium, IBARAKI University
5) Speed Up for Large Scale Simulation by SPINTO, ASMO



2.2.2 Shanghai Feiyi

Shanghai Feiyi held PHOENICS User Seminars in March and April (details below). All seminars were free of charge and any interested parties were invited to attend. For information regarding future dates and details please contact:

Miss Niu,	Shanghai Feiyi Software Technology Ltd.
Address:	Room 1916-1917, 81# South Qingzhou Rd,
Tel:	021-64821011，64822252
Email:	support@shanghaifeiyi.cn

Topic:	Two Fluid Model
Address:	Institute of Process Engineering, Zhongguancun, Beijing
Topic:	Architecture application
Address:	Architecture Dept, Tsinghua University
Topic:	Architecture application
Address:	Architecture Department, South China University of Technology. Guangzhou
Topic:	PHOENICS training
Address:	Shenzhen Institute of Building Research, Shenzhen City

To contribute to this Newsletter please email articles to cik@cham.co.uk. Thank you.

3) PHOENICS Applications

3.1 CFD for an Animal Environment

Brian Spalding, CHAM Limited, March 2011

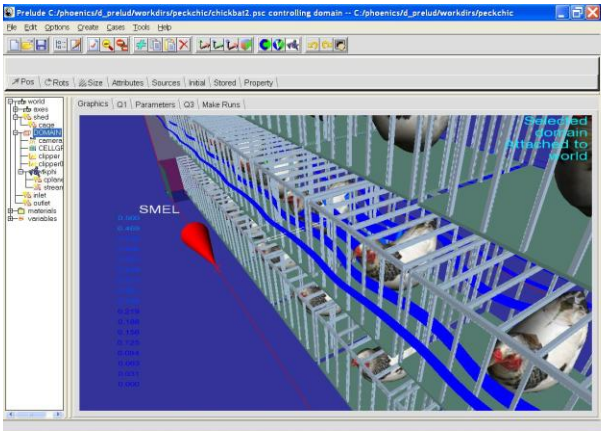
This article arose from the posing of a problem regarding flow of air, and temperature, in an animal environment, in this case a battery-hen house. The hen house was regarded as a typical environment in which air temperature and composition distributions in space and time were predicted.

Input data included geometry (what is where); production rates of heat and pollutant from each cage; air-supply rate and temperature. Free (gravity-induced) convection was taken to be as important as forced.

The problem does not contain complicating factors such as multi-phase, chemical reaction or compressibility. However, the turbulence is low-Reynolds number and gravity-influenced; and radiation is not insignificant.

Specialists in the animal-production-environment know little about CFD; therefore easy-to-use programs with simple data input methods save time and minimise mistakes. Simulation results should be presented so they are easy to inspect and interpret.

The problem fitted the CHAM CFD-Gateway concept of PHOENICS which enables users to access only the items needed for a particular interest and which can use only the terminology appropriate to that interest.

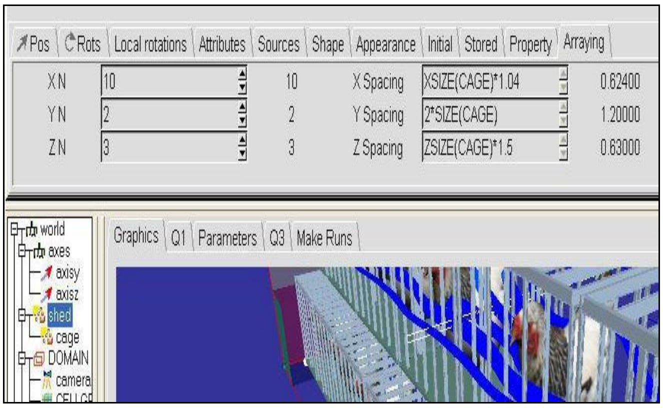


A Gateway relating to CFD for hen houses could look as per the menu panel above with part of a chicken-battery building in its graphics window.

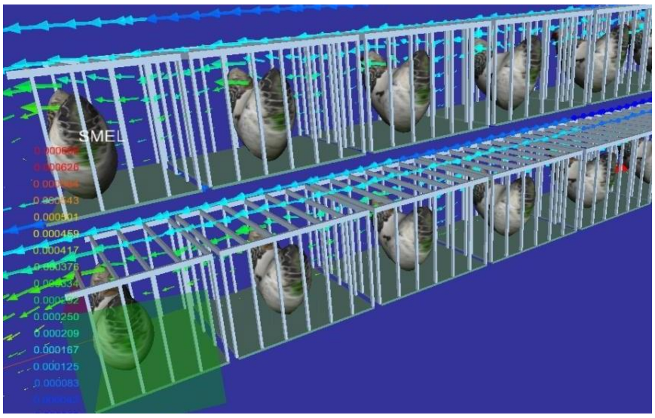
To create a new gateway, one need not start from scratch; for the PRELUDE package exists, with a ready-to- use general framework and customising tools which provide the frequently-required functions of opening cases, introducing, modifying or deleting objects, running simulation calculations etc.

An object tree can be created for a hen-house gateway where clicking on an item enables its attributes to be reviewed and edited. These items control results-display items, *viz* stream-lines; and 'cutting planes' for contours and vectors; and the in-and out-flow of ventilating air. Variables can be stored and solved for and specialist items (such as SMEL of which chickens are a source) can be amongst them.

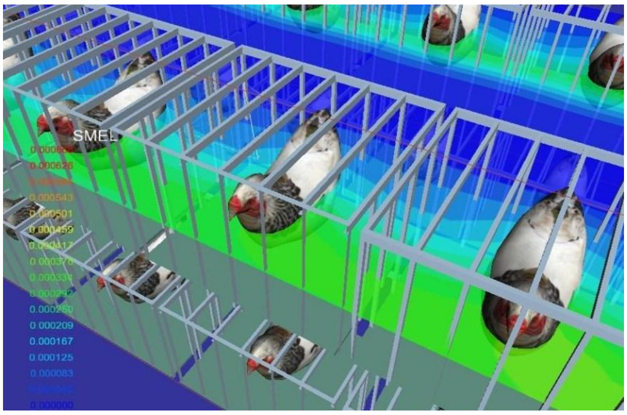
It would be tedious to introduce cages one by one so PRELUDE offers an array facility seen below:



After a user in satisfied with input data the run button launches the simulation and results can be displayed as velocity vectors (below):



Results can also be displayed as contour displays (below) in accordance with pre-set orders, or the user's menu-expressed wishes. The picture below also shows the SMEL concentration:



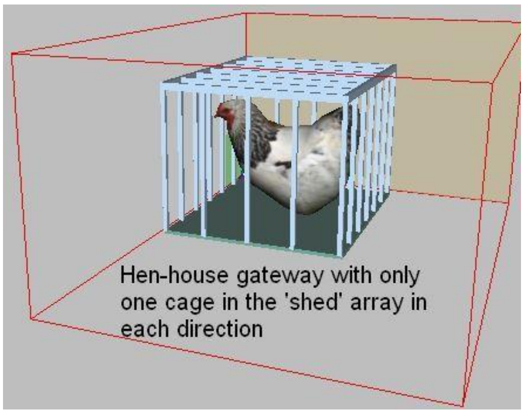
The Gateway shown here was created in a few days; it would take longer to make it ready to accept the data which a hen-house designer might specify such as:

- O cage dimensions
- O rates of production of heat, odour, etc per hen
- O locations and power of fans, heaters and coolers
- O external atmosphere conditions

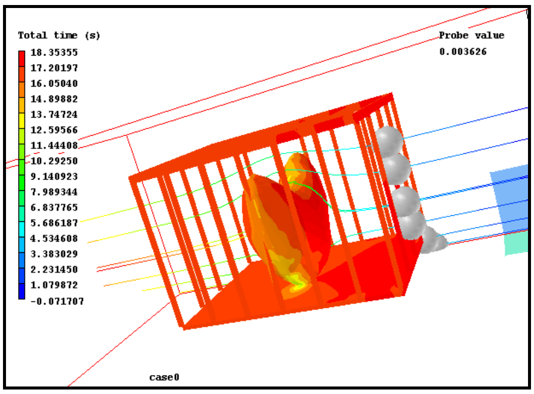
Also the CFD specialist should optimise

- O models of turbulence and radiation, and
- O solution control settings.

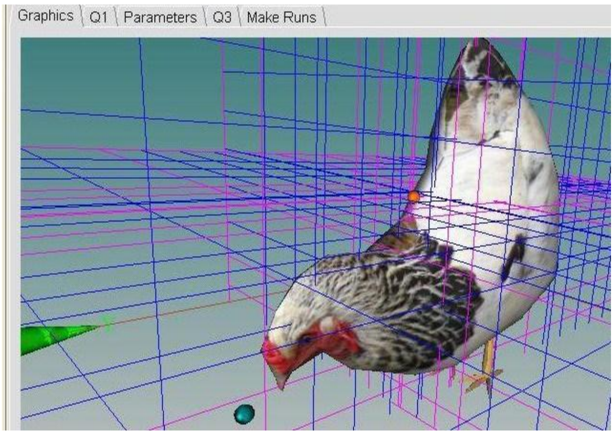
A combination of space-averaged and detailed-geometry CFD can be applied to hen-house simulation by setting array parameters to 1,1,1 so that all computational cells can be concentrated in the smaller domain surrounding a single cage. This means that transfers of heat, matter and momentum from hen to air can be more accurately simulated. Doing this at various Reynolds numbers and wind angles allows formulae to be developed which can be used in whole-hen-house space averaged-CFD simulations.



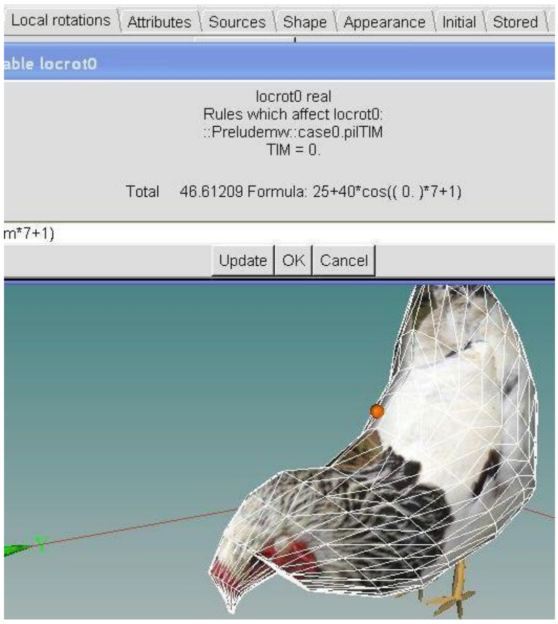
Flow around one chicken:



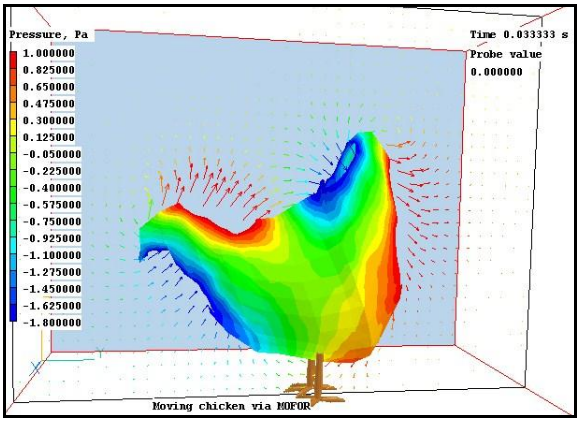
PRELUDE can be used for moving objects. Here is a chicken in a cage which used a detailed-geometry grid set up for a time-dependent simulation.



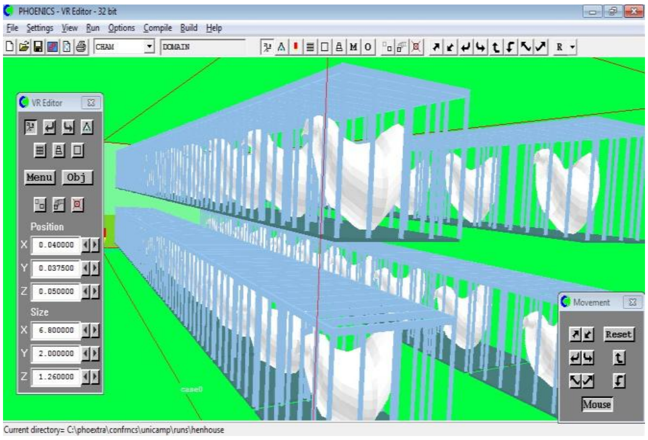
PRELUDE enables the motion of the chicken to be prescribed via formulae including one which could describe its periodic pecking motion as shown below which, on a web site, is animated.



The following shows results of a simulation including air-velocity vectors and surface-pressures based on detailed-geometry CFD. Hen-house designers need not use such detail.



The PHOENICS VR Editor provides a graphical user interface which facilitates the setting up of single-instance flow-simulation as shown below. This is convenient if only a single scenario is in question.



CFD Online

A free online centre for Computational Fluid Dynamics. Share news and experiences with other PHOENICS Users at www.cfd-online.com/forums/phoenics.

4) User Applications

4.1 Fluid Dynamic Models for Hood Efficiency

by Giacomo Ferrarese and Marco Maria Agostino Rossi
Politecnico di Milano Univeristy

Introduction

We refer to a thesis on the efficiency of home suction devices; the work was conducted at the IIAR dept. of the Politecnico di Milano University. PHOENICS 2009 was used to provide numerical simulations of different hood models placed in a standard room (figure 1) sized according to European standard IEC 61591. Because of symmetry we needed to study only half the room.

The first part of the work defined numerical parameters and dimensions of the discretized grid to study problem [2-3] as a basic case. For the most part of this research, we consider a simplified model where the density ρ of the fluid in the domain and the temperature are constant. Then we improve our model, introducing a temperature variable. The turbulence model, chosen by literature [4-5], is the K-ε model.

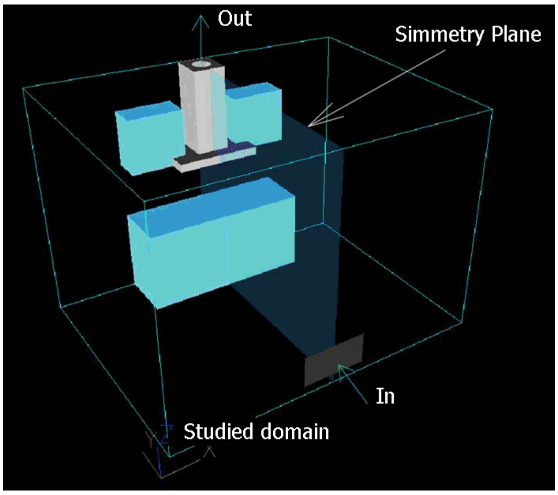


Figure 1: Standard kitchen according to IEC61591

The numerical solution is calculated by an elliptic-staggered scheme. We set a grid able to guarantee a stable solution with the lowest possible computational cost. Connected to the grid dimension, the number of iterations of the calculation was defined.

To this end, we took nine reference points within the domain, in which the modules of speeds and pressures are monitored. We assessed the convergence of the solution, changing grids and iterations.

Six different steps of refinement for the grids were considered. They are referred to as cases Z,Y,A,B,C,D and are ordered from least to most accurate. Figure 2 shows an example of convergence in the domain. In abscissa are the number of iterations and in the ordinate there are the errors with respect to the reference solution, which was taken for the grid with a higher definition for 5000 iterations.

Each curve shows the results for a different configuration of the grid. This analysis led to the selection of a grid-by-step average 3cm (case B) and 2000 computation iterations.

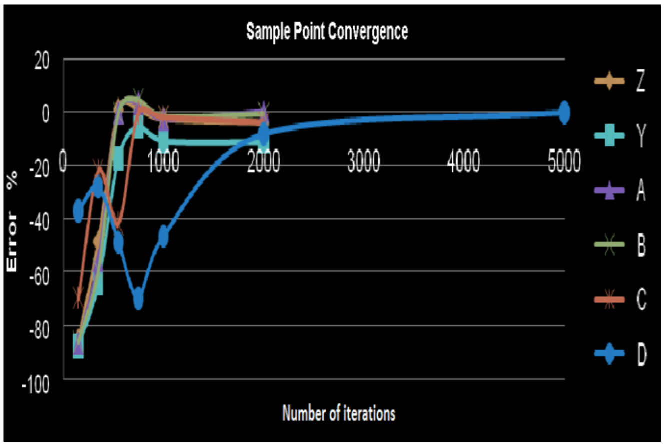


Figure 2: Relative error in function of the resolution of the grid and iterations number

Discussion of Results

Because is not very clear how to define the efficiency of a hood, and because of the simplified model, the analysis of simulations was carried out by various methods of which we report the most significant. Figure 3 shows three of the hoods considered in our work.

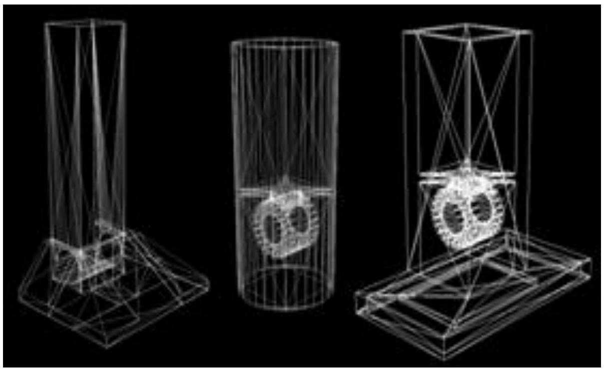


Figure 3: Chimney, Circular , Standard

The first method is based on the introduction of four horizontal planes. Figure 4 shows the planes placed between the stove top and the mouth of the hood through which only upward flow is considered. Results are reported in Figure 5.

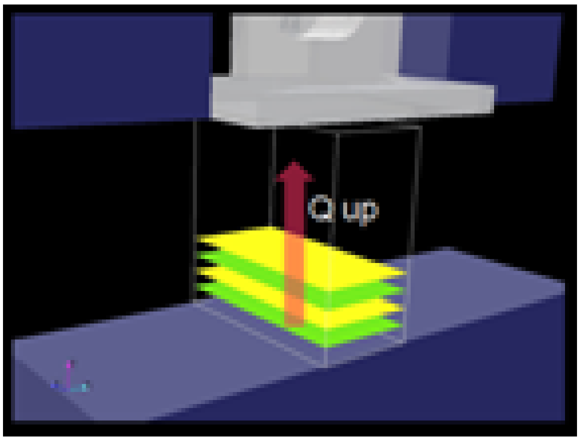


Figure 4: Sample planes

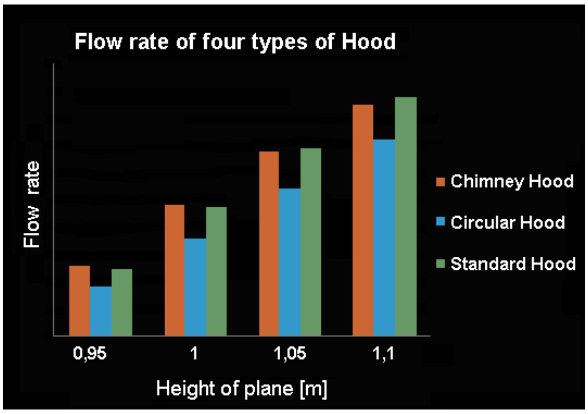


Figure 5: Flow rate through the sample planes for the 4 hood models

The behavior of three different types of hoods is different with varied ventilation capacities. The second method is based on the introduction of a control volume placed between the stove top and the mouth of the hood, as reported in figure 6.

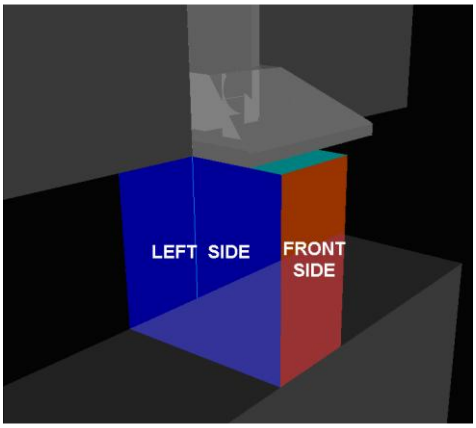


Figure 6: Control Volume Position

We discretized volume control surfaces in sub-areas. For example the left side was divided by three columns and six rows. This operation shows local suction behavior then we analyzed the flow passing through these areas. We report the results in figure 7; here the surfaces are colored by the modules of normal velocity.

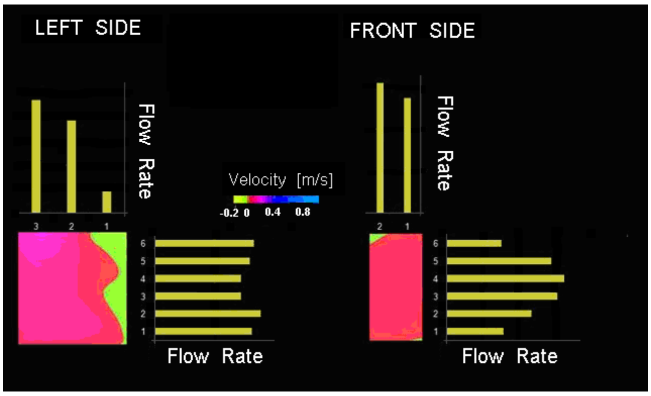


Figure 7: Flow rates through control volume

Further developments have been elaborated by introducing a variable temperature to the problem. This change helps to create a realistic model more accordant to the European standards. The analysis of preliminary result with the viewing of the streamlines showed that the introduction of a hot plate at 200° C radically alters the results. In fact, as shown in picture 8, there is a wide dispersion of the flows underlying the hood, which previously did not exist.

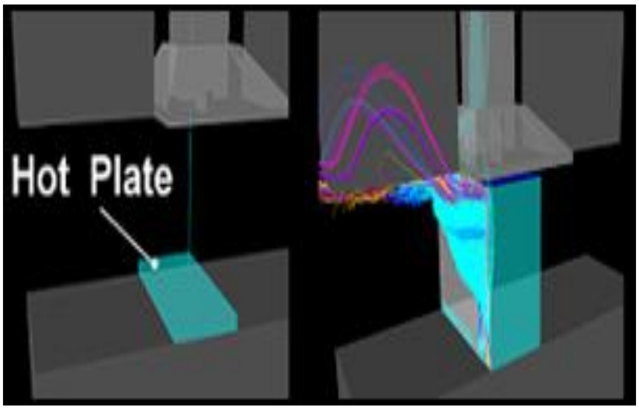


Figure 8 : Improving the model with an heat source and streamline due to the new conditions

Difference in domain geometry could introduce significant changes to the solution. To study more realistic conditions we decided to consider different realistic variations of the geometry such as the presence of doors, or a person, in the room (see figure 9).

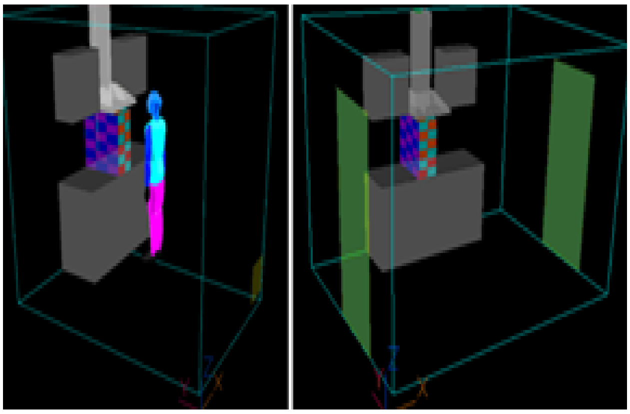


Figure 9: New geometry conditions imposed

Preliminary results of the latter case are in figure 10.

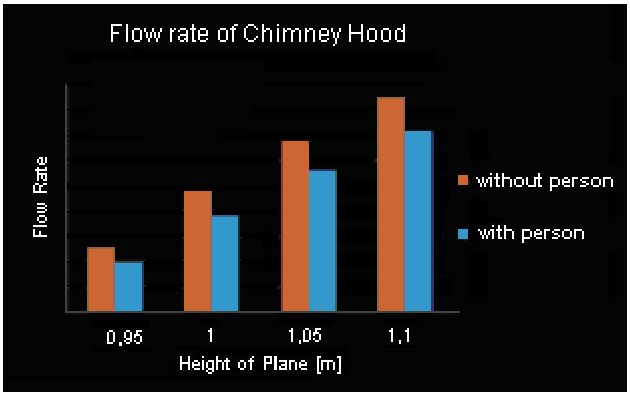


Figure 10: Comparing flow rate through the sample planes for conditions with and without human presence

Conclusions

The simple numerical model used to analyse the behavior of hoods under different fluid dynamic and geometry conditions was able to reach stable results and good convergence. Comparing the numerical simulation with different post processing methods, we defined different indexes to evaluate the hood efficiency.

We considered possible improvements to the numerical model for a more realistic solution. By improving the model it would be possible to study more complex configurations of room and boundary conditions.

References

1] "Modellazione fluidodinamica e analisi dell'efficienza per le cappe di aspirazione" thesis by Giacomo Ferrarese & Marco Maria Agostino Rossi, 2010. Tutor: Professor S Malavasi,.

2] "Fluidodinamica computazionale" by Peter V. Nielsen, Francis Allard, Hazim B. Awbi, Lars Davidson, Alois Schälin.

3] "Measurement and prediction of indoor airflow in a model room" by J.D. Posner, C.R. Buchman.

4] "Simulation with different turbulence models in an annex 20 room benchmark test using Ansys CFX 11.0" by Li Rong, Peter V. Nielsen.

5] "Comparison of Turbulence Models for Numerical calculation of Airflow in an annex 20 Room" by M.Sc. Lars Køllgaard Voigt.

4.2 Turbulent Flow Around a Semi-Cylindrical Obstacle

Students of Professor Stefano Malavasi, Politecnico di Milano.
email: stefano.malavasi@polimi.it.

Extract from Meccanica dei Fluidi.
www.diiar.polimi.it/idra/corso.asp?id=95#Progetti

Introduction

In this work the effects of the presence of a semi-cylindrical obstacle, immersed in a turbulent flow, are investigated. The flow is confined to a rectangular section channel of fixed size, and the obstacle is placed in the cross-flow direction as will be clarified below.

Aims

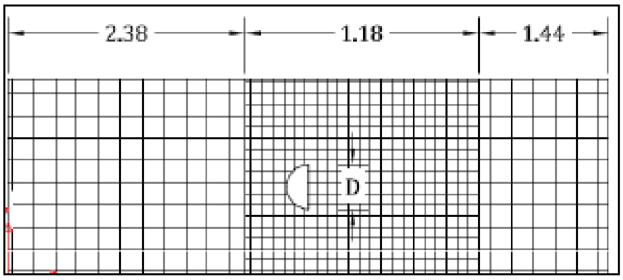
- Determine the vortex detachment frequency versus one of the two characteristic sizes of the obstacle.
- Investigate location of the stagnation point versus the obstacle height from the bottom of the channel.
- Evaluate the drag and lift coefficients varying both the obstacle location and its shape ratio L/D.

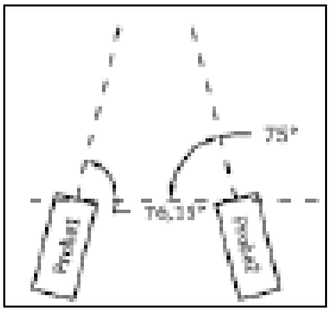
WORK STEPS

- Experimental analysis:
  - Processing of velocity data obtained from some experimental sessions, to get velocity profiles.
  - Frequency analysis of the same velocity data to evaluate empirical vortex detachment frequency and, consequently, the Strouhal number.
- Numeric analysis:
  - Evaluation of velocity profiles & relative Strouhal numbers to compare with experimental data.
  - Investigation of the flow around the obstacle through numerical simulation.

EXPERIMENTAL APPARATUS

- Channel:
  - Length L = 5m.
  - Width B=0,5m
  - Slope i=0,00%
- Obstacle:
  - Diameter D = 0,06m
  - Width B=0,5m
  - Position along x axis x=2,5m
  - Position along y axis y=0,14m
- Echo – Doppler velocimeter:
  - 2 probes located at the bottom of the channel with an inclination angle as in the figure below:





EXPERIMENTAL ANALYSIS

- Empirical phase conditions were:
  - Fixed location for the obstacle, 2.5m along x and 0.14m from the channel bottom.
  - 3 independent experimental sessions with slightly different boundary conditions (see table).
  - 3 different probe locations, re the obstacle, for each session: 4.5, 18 and 36 cm downstream.

	Vm [m/s]	Qm [l/s]	Hm [m]
Session 1	0.298	59.5	0.4
Session 2	0.350	70.0	0.4
Session 3	0.250	49.9	0.4

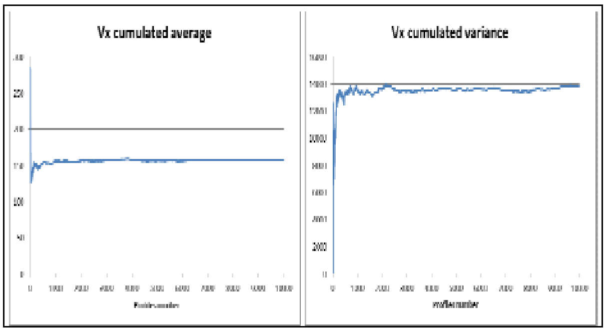
EXPERIMENTAL ANALYSIS

- Two echo-doppler provide instantaneous velocity values along the rays' direction, which, combined, give velocity values along the x and y axis.
- Since this evaluation is exact only in the crossing point of the two rays, the obstacle was located at the same height (17.1 cm) of this point.
- The echo-doppler was operated as follows:
  - Sampling frequency 25.64hz
    (1 profile every 0.039 s)
  - 10'000 profiles sampled for each session.

EXPERIMENTAL ANALYSIS

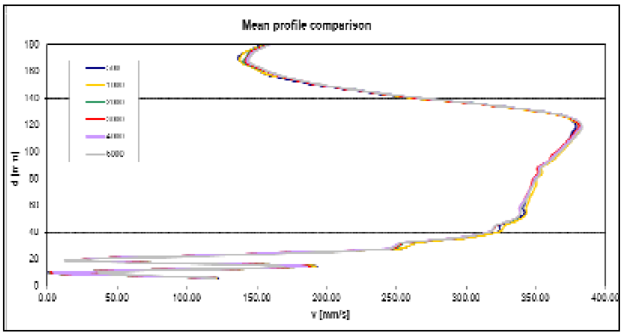
Since the investigated flow is turbulent ($Re \approx 10^5$) the number of profiles sampled must be sufficient to ensure a correct average velocity value.

As the cumulated average and the cumulated variance of Vx tend to a finite value the profiles number is correct.



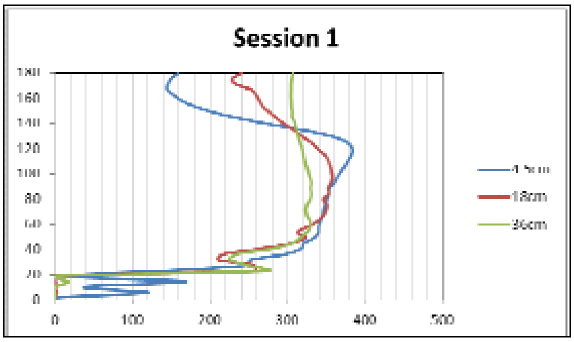
EXPERIMENTAL ANALYSIS

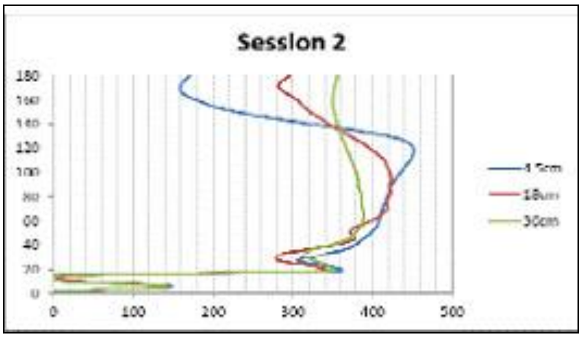
As can be noted from the cumulative average and variance graphs they tend to a mean value for lower than 10'000 profile number. This fact is confirmed by the following figure in which mean velocity profiles for different number of measurements are compared:

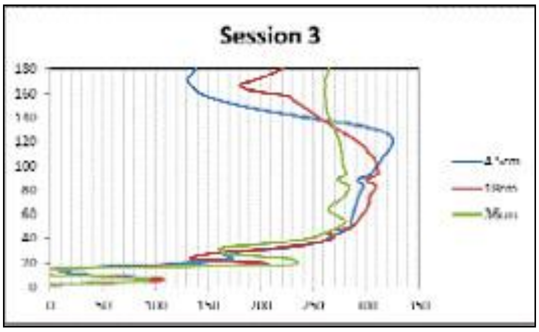


EXPERIMENTAL ANALYSIS

- Vx velocity profiles for 3 different sessions, and at 3 different probes location downstream.
- As can be seen from the graphs, the more downstream the probes' location, the less intense the wake effect due to the obstacle (see y=170mm).







NUMERICAL ANALYSIS

- Numerical analysis was carried out using the finite-volume software PHOENICS.
- Boundary conditions were the same as the first experimental session.
- Some hypothesis were introduced to simplify the analysis and to reduce its computational weight:
  - The velocity field is symmetric on the z=0.25m plane so analysis can be bi-dimensional.
  - Mach number Ma<0.3 for our operating velocities, so water compressibility is negligible.
  - There are no different fluid interfaces so the dependence on the Weber number can be considered zero.
  - Froude number has not been considered since the PHOENICS only considers confined surface flows with pressure set as p=patm.
- An initial steady-flow simulation phase showed that, since the phenomenon is unsteady, a time-transient flow analysis was necessary.
- A preliminary phase was performed to set up simulation parameters, like mesh refinement, and time steps, for the transient analysis.

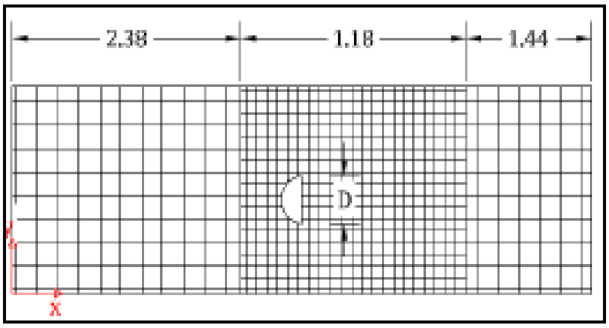
NUMERICAL ANALYSIS

- Parameters set up: Time: t_0= 0 s, t_{fin}= 10 s
  Time steps number = 1000 that leads to f=100Hz.
- Iteration number = 15 for the transient simulation.
- Dumping: Number of dumping points=100 that leads to 10 Hz. The experimental sampling frequency was equal to 25.6 Hz.

NUMERICAL ANALYSIS

- Mesh set up:

The domain was divided into 3 regions along x axis; a central one, called nil object, contains the obstacle and has a much finer mesh. In the y direction one region was defined, and in z direction only one cell for symmetry.



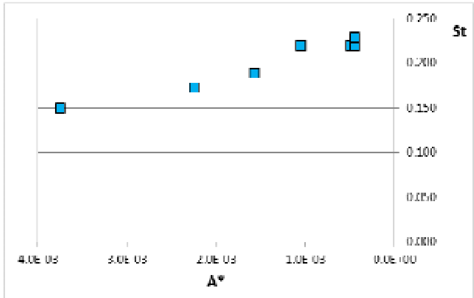
NUMERICAL ANALYSIS

- The cell number in each region and every direction was defined for an increasingly refined mesh series. The parameter representing the mesh accuracy is: $A^*=A_{cell}/A_{obst}$ where A_{cell} is the area of the nil object cell, and A_{obst} is the obstacle surface projected on zy plane.
- The meshes chosen are the following:

MESH	X1	X2	X3	Y	Z
GT1	20	70	30	30	1
GT2	20	70	30	50	1
GT3	20	100	40	50	1
GT4	20	150	50	50	1
GT5	20	200	50	80	1
GT6	20	220	50	80	1
GT7	40	220	60	80	1
GT8	50	240	60	110	1

NUMERICAL ANALYSIS

- **Velocity data frequency analysis:**
- **Since the Strouhal number is defined as $St = \frac{D \cdot f}{U}$ where f is the spectrum frequency of the v_y velocity a frequency analysis was performed for each mesh in order to get the following graph:**

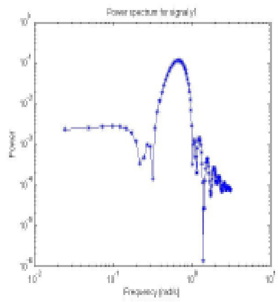


NUMERICAL ANALYSIS

- **As can be seen from the previous graph the Strouhal number tends to a finite value ($St \cong 0.22$) after mesh number GT5. This mesh will be used for the numeric simulation of the problem.**
- **The Strouhal number coming from the numeric analysis has been compared to the ones from experimental data, leading to the result:**
- **$St_{exp} \cong 0.393$**
- **$St_{num} \cong 0.220$**

Different facts can be responsible for this difference:

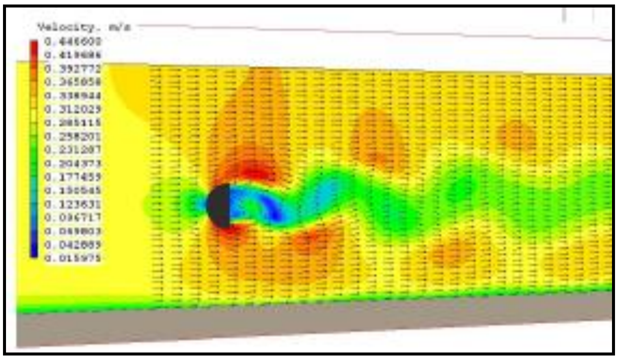
- The channel lateral walls are not considered in the model.
- Free surface flow in reality, not in the model.
- Uniform inlet velocity in the model.

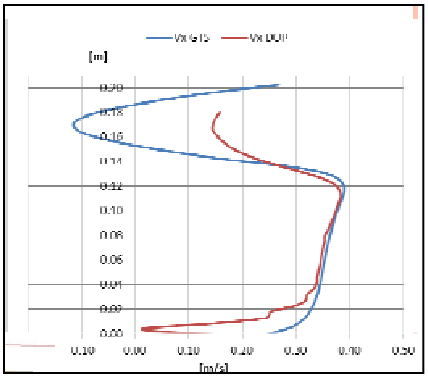


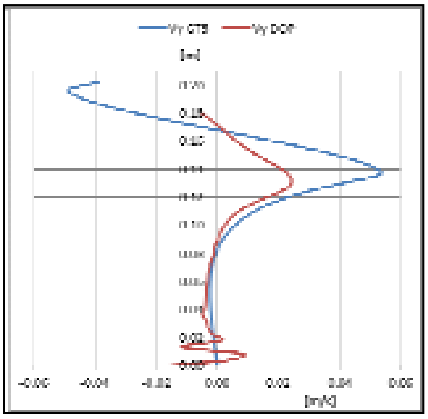
V_y spectrum. Numeric data.

NUMERICAL ANALYSIS

- Mean Vx and Vy velocity profile comparison between experimental and numerical data:
- Experimental data are relative to session 1. Time averaged numerical data relate to GT5 mesh.

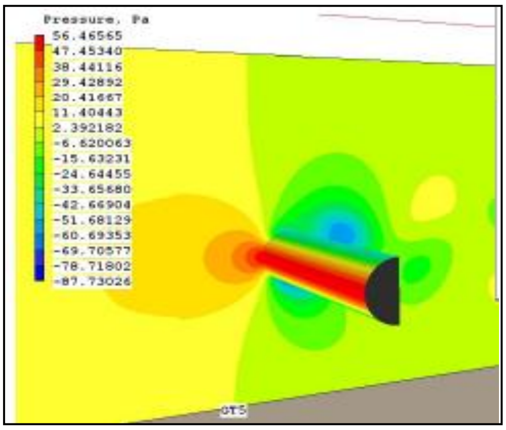


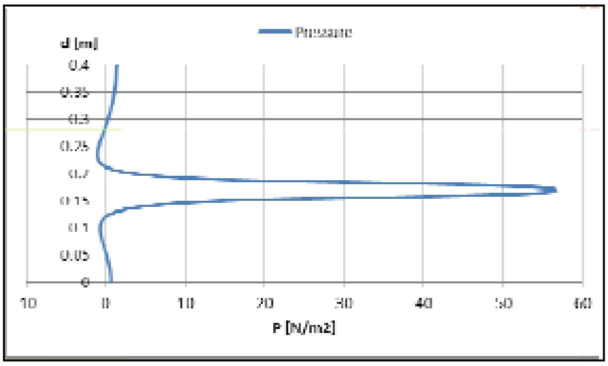




NUMERICAL ANALYSIS

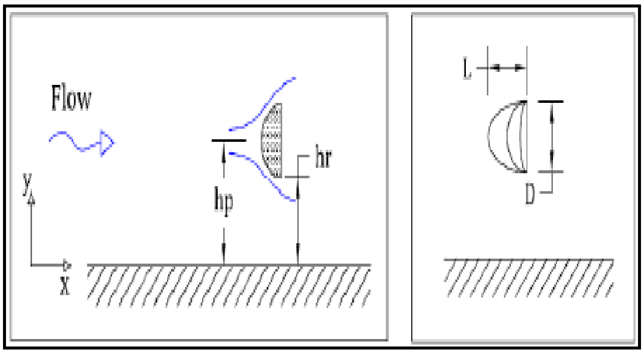
- Pressure values relative to GT5 mesh.
- Pressure reaches maximum value at the stagnation point, corresponding to the minimum velocity value.





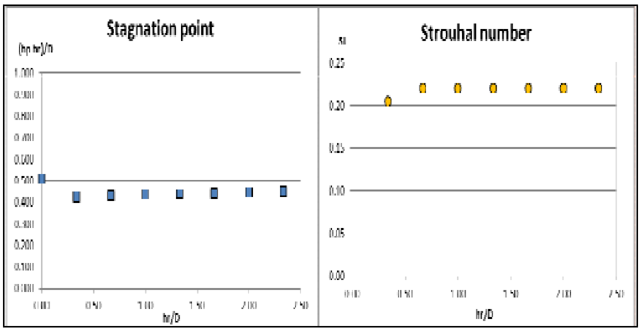
NUMERICAL ANALYSIS

- The previous comparisons are useful to assess that the numeric model gives a sufficient agreement with the experimental data; so it is possible now to investigate – just with the numeric model – other configurations of the system, and its response.
- Variation of the obstacle height (h_r) from the bottom of the channel.
- Obstacle shape ratio variation (L/D).



NUMERICAL ANALYSIS

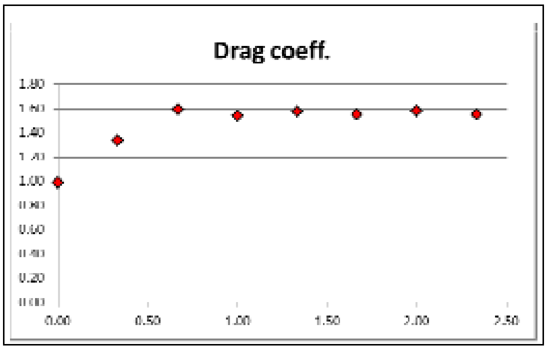
- Obstacle height (h_r) variation.
- Stagnation point height (h_p), Strouhal number and Drag coefficient investigation.



Except for the first point (h_r=0) a slight upward movement is observed for the stagnation point. Strouhal number variations are observed only very close to the bottom.

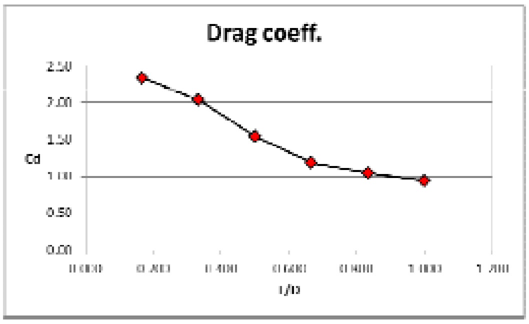
NUMERICAL ANALYSIS

- Drag coefficient variation versus obstacle height.
- A significant decrease is observed close to the bottom wall; in the rest of the field the drag coefficient is almost constant.



NUMERICAL ANALYSIS

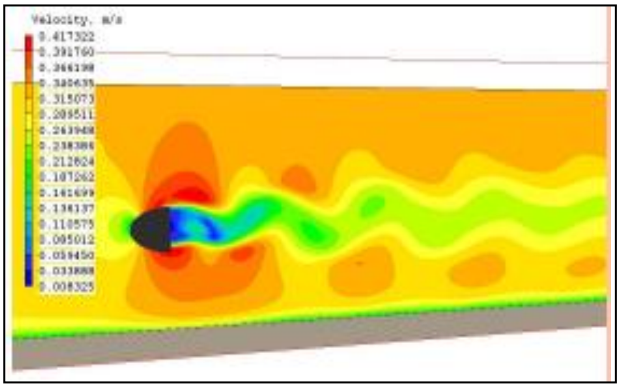
- Obstacle shape ratio variation (L/D).
- Strouhal number, Drag and Lift coefficient investigation.

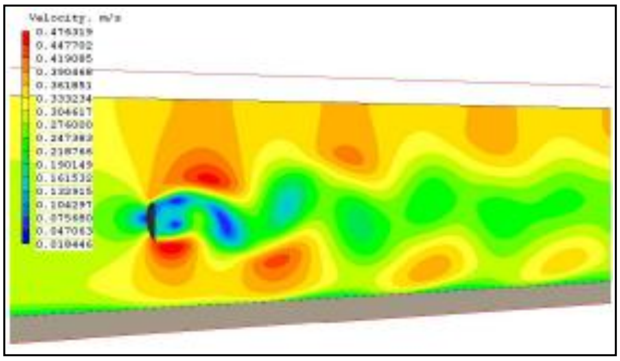


- A decrease of the Drag coefficient occurs increasing L/D as one might expect. This is in good agreement with the wake variation effect shown below.

NUMERICAL ANALYSIS

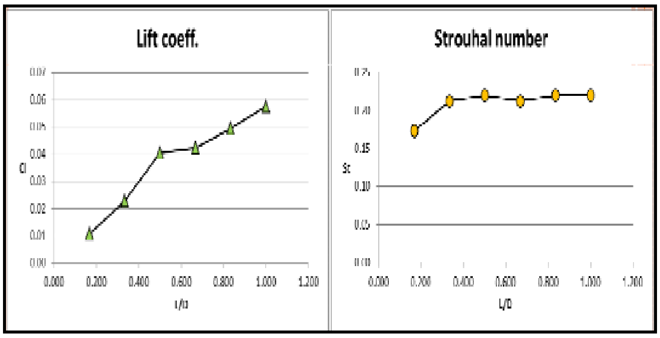
- Wake effect for 2 different L/D shape ratio values: 0.167 and 1.0 respectively.
- A difference in the vortex creation is noticeable between the two cases.





NUMERICAL ANALYSIS

- For increasing L/D ratios an increase in the Lift coefficient is observed.
- A slight decrease of the Strouhal number occurs just for small shape ratios, being almost constant for the other conditions.



RESULTS AND COMMENTS

- The results of the numeric simulations and the experimental data are qualitatively in good agreement and the mean velocity profiles correspond also quantitatively.
- The drag coefficient obtained is similar those known in literature for the same shape.
- A difference in value is observed for the Strouhal number.
- The differences can be due to more than one reason:
- The channel lateral walls are not considered in the model.
- Free surface flow in reality, not in the model.
- Uniform inlet velocity in the model.
- 2D analysis to decrease the computational cost.
- Default friction boundary conditions as input.

Students of Professor Stefano Malavasi, Politecnico di Milano. email: stefano.malavasi@polimi.it. Extract from Meccanica dei Fluidi.www.diiar.polimi.it/idra/corso.asp?id=95#Progetti

STOP PRESS

Professor Spalding is honoured to have been awarded the 'Huw Edwards Award' for lifetime service and contribution to Combustion Physics. He is the fifth recipient of the Award which is conferred by the Institute of Physics, Combustion Physics Group.