PHOENICS News

Table of Contents

Table of Contents	Page Number
Numerical modelling of the turbulent flow in a pipe bend, by Qi Yang and Gianandrea Vittorio Messa, "Fluid Lab" research group, Politecnico di Milano	2
Analysis and modelling of the reaction-mixture interaction on the extrapolation of the aerobic and anaerobic fermenters, by Amaury Danican, Jean-Pierre Fontaine, Christophe Vial , Université Clermont Auvergne, France	4
PHOENICS FLAIR – Modelling Aerosol Deposition in Buildings By Harry Claydon and Mike Malin	5
News from CHAM Agents – Shanghai Feiyi	6
News from CHAM Agents – ArcoFluid	7
News from CHAM Japan	8
News from CHAM	8
Contact Us	8

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

Autumn 2019

Pipe bends or elbows are inevitably found in many industrial applications, including the slurry pipeline systems used in the mining industry to transport mineral concentrate, extracted from a mine, to the mining processing plant. Turbulent flow in a pipe bend is more complicated than that within a straight pipe due to flow separation and secondary motions induced by the actions of centrifugal force and radial pressure gradient.

Computational fluid dynamics (CFD) allows deep insight into the analysis of the complex physical mechanisms occurring inside a pipe bend and, compared to costly and time-consuming experimental tests, is an attractive investigation approach. As the first step in the development of his PhD Thesis at Politecnico di Milano, Qi Yang, supervised by Dr. Gianandrea Vittorio Messa, used PHOENICS to simulate the turbulent single-phase flow in a pipe bend by reproducing experiments reported in the literature. Indeed, a lot of experimental and modelling work has been done on this type of flow in recent decades. In this study, reference was made to the experimental research of Sudo et al. (1998), who tested a 90° pipe bend with diameter d=0.104 m, curvature radius R_c=0.208 m and with upstream and downstream pipe lengths of 3.12 (z'/d=-30) and 1.56 m (z/d=15). The working fluid was air, with density 1.19 kg/m^3 and kinematic viscosity 1.54×10^{-5} m^2/s flowing at pipe-bulk mean velocity of U_{in}=8.7 m/s, resulting in a Reynolds number equal to 6×10^4. The flow is characterized by the Reynolds number and the curvature ratio, $\gamma = D/2R_c$ (here γ is 0.25), and is sometimes expressed in terms of the Dean number,$De = Re\sqrt{\gamma}$, which here is 3×10^4. A sketch of the bend is shown in Fig. 1, together with the coordinate system adopted.

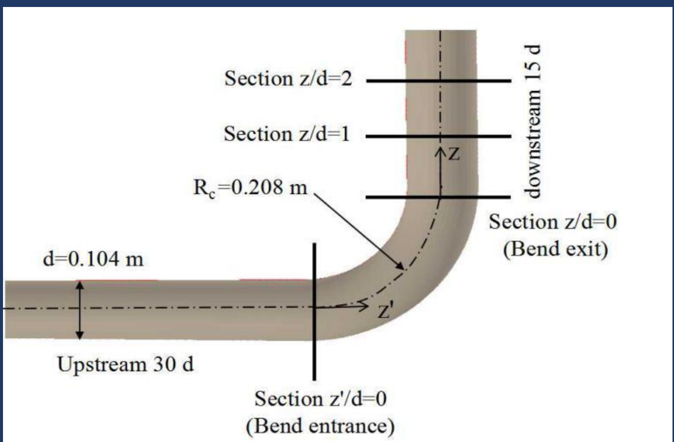


Fig. 1. A sketch of the pipe bend and coordinate system.

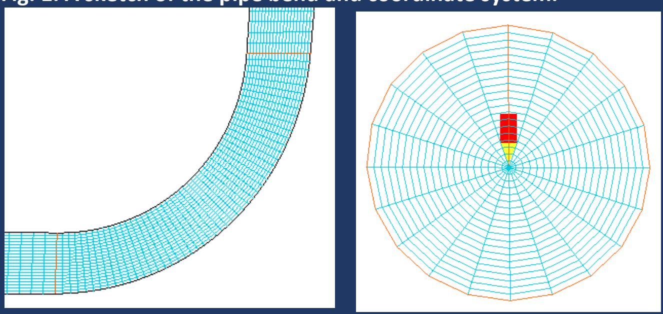


Fig. 2. Discretization of the pipe bend geometry: side view (left) and front view (right)

The computational domain of the PHOENICS simulations comprised the bend and two adjacent straight pipes, whose lengths are equal to 30d and 15d, respectively. The upstream pipe bend is sufficiently long to achieve fully developed flow at the bend inlet, thereby allowing meaningful comparison with the experiments of Sudo et al. (1998). Although a plane of symmetry exists in the single-phase flow solution, it was an intentional decision to simulate the entire pipe section. This is because the current study is a preliminary study towards the simulation of the two-phase slurry flow inside a pipe bend in the horizontal plane, which is no longer symmetrical due to the effect of gravity. The structured mesh generated in PHOENICS using Body Fitted Coordinates (BFC), partially sketched in Fig. 2, was suitable to obtain a grid-independent solution. The steady-state Reynolds equations coupled with the standard k-ε turbulence model were solved in the "Elliptic-Staggered" formulation. Cyclic boundary conditions were applied to all slabs and the "symmetry treatment" option was activated for an accurate evaluation of the convection fluxes at the faces of the finite-volume cells. Preliminary results are shown in Figs. 3 and 4. Particularly, the contours in Fig. 3 give a clear idea of the spatial evolution of the mean axial velocity inside the bend. The "symmetry treatment" option allows obtaining a symmetrical solution, in agreement with the experimental observations of Sudo et al. (1998). Figure 4 compares the axial velocity profiles along the symmetry diameter at the entrance and exit sections of the bend with the experimental data of those researchers. In the present study, no flow separation is observed on the inner pipe bend surface in either the CFD results or the experiment. This is to be expected as, according to literature (Weske, 1948; Hellstrom et al., 2011), flow separation is only likely to occur when γ is greater than 1/3. The CFD predictions are in good agreement with the experiments in the entrance section and at the extrados (outer concave surface) of the exit section. Conversely, the numerical simulation tends to underestimate the mean axial velocity at the intrados (inner convex surface) of the exit section. This is a critical region of the flow, especially in the cases when flow separation occurs. The next phase of this research will be investigating the influence of different turbulence models and wall treatments. Once the computational model is finalized for the single-phase flow, it will be possible to start addressing the more complex two-phase slurry flow case.

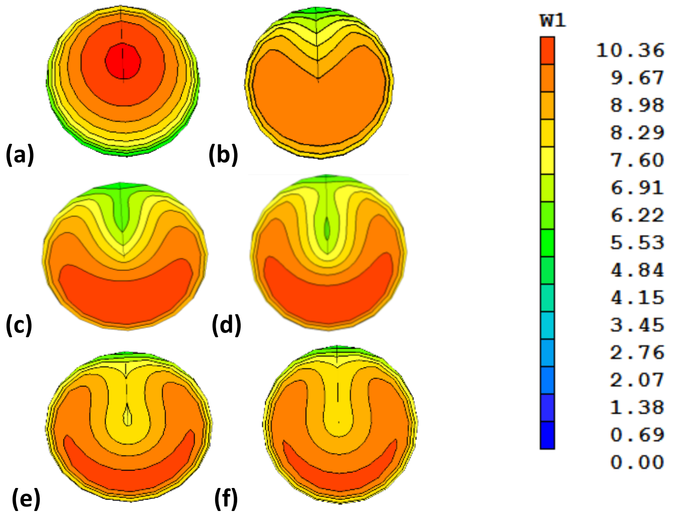


Fig. 3. Contours of the mean axial velocity at different cross sections: (a) bend entrance; (b) bend exit; (c) z/d =1; (d) z/d=2; (e) z/d=5; (f) z/d=10.

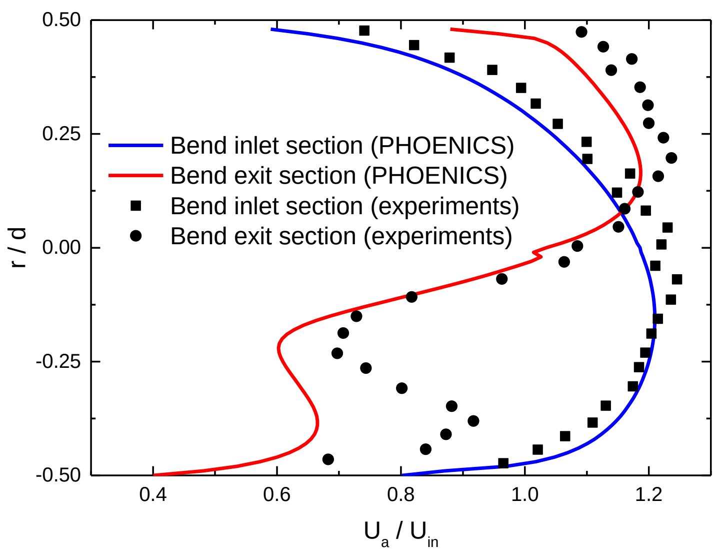


Fig. 4. Mean axial velocity profile along the symmetry diameter at the entrance and exit sections of the pipe bend: comparison between the PHOENICS results and the experimental measurements of Sudo et al. (1998).

References

Sudo, K., Sumida, M., & Hibara, H. (1998). Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend. Experiments in Fluids, 25(1), 42-49.

Weske, J. R., (1948). Experimental investigation of velocity distributions of downstream of single duct bends. N.A.C.A. TN1471, 1471.

Hellström, L. H. O., Sinha, A., & Smits, A. J., (2011). Visualizing the very-large-scale motions in turbulent pipe flow. Phys. Fluids,

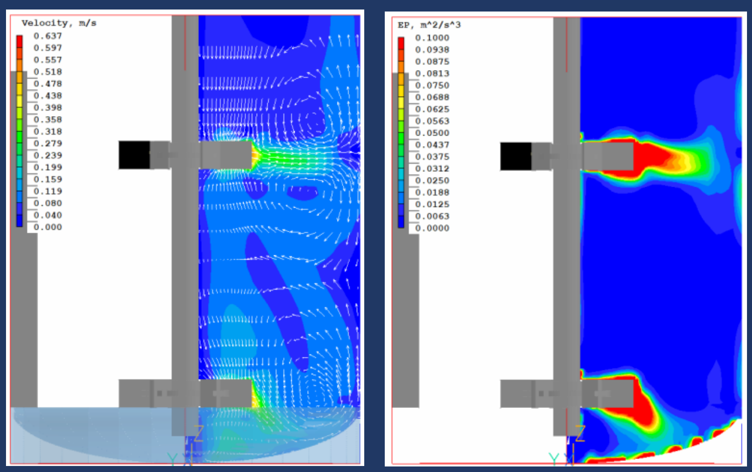
Analysis and modelling of the reaction-mixture interaction on the extrapolation of the aerobic and anaerobic fermenters, by Amaury Danican, Jean-Pierre Fontaine, Christophe Vial, 1) Université Clermont Auvergne, France

The "dark fermentation" process consists in the decomposition of organic substrates thanks to microorganisms in agitated bioreactors, also called fermenters, to produce biohydrogen and other molecules of commercial interest. It is a well-known process but still needs to be optimized. An optimum process needs a good homogeneity of the reactional environment while limiting the energy consumption by the impellers. Hence, it is necessary better to understand the hydrodynamics and the concomitant mass transfer within the fermenter in order to select the best operating conditions of the bioreactor and their impact on the mixture. [1][2]

The main objective of this work is to develop a three-dimensional CFD model using PHOENICS®, which can describe the behavior of the liquid within the reactor including all of the involved physical phenomena. These parameters are compared to the results obtained in a real fermenter in order to analyze their effects on the process, which will lead to the optimization of the production of bioH2.

The first step was to simulate a single phase flow of a Newtonian fluid in a steady state. The number of cells was tested as well as the turbulence models and the discretization schemes. The results were compared to the experimental measurements of 2D-PIV (Particle Image Velocimetry) to choose the best parameters and thus to check the reliability of the model. The mesh count of 144 x 63 x 74 cells was retained as well as the Kao-Launder model and the VLAN1 scheme. The power consumption, calculated thanks to the momentum of the forces on the impellers, was also compared to the literature data. Afterwards, the model was used for Non-Newtonian fluids. The Cross model, describing the rheological behavior of the shear-thinning fluids, was entered as an In-form formula to modify the viscosity in each mesh following the shear rate calculated.

(a) (b)



Flow structure: Velocity (t) & EP (r) in a bioreactor for "dark fermentation"

The CFD results gave us access to the structure of the flow in different mixing conditions (fig 1a.). Moreover, it was possible to extract the areas of high stress for the microorganisms due to the turbulence, using the local rate of dissipation of turbulent kinetic energy ε (fig 1b.). This value enables the calculation of the Kolmogorov length scale, representing the size of the smallest turbulent eddies. These eddies can destroy the aggregates of microorganisms growing in the bioreactor and thus inhibit the reaction producing the bioH2 [3]. It is now possible for rotation speeds, viscosities or even for several designs of impeller and reactors, to check the hydromechanical stress on the microorganisms, allowing us to choose the best parameters to optimize the process. The next step will be to simulate a two-phase flow using the IPSA solver of PHOENICS® to study the effect of the gas on the flow structure and the gas-liquid transfer.

References

[1] Trad Z., Vial Ch., Fontaine J.-P., Larroche Ch. (2017). Mixing and liquid-to-gas mass transfer under digester operating conditions, Chemical Engineering Science, 170, 606–627.

[2] Chezeau B., Vial C. (2018) Combined effects of digestate viscosity and agitation conditions on the fermentative biohydrogen production, Biochemical Engineering Journal.

[3] Chezeau, B., et al. (2020) Characterization of the Local Hydromechanical Stress through Experimental and Numerical Analysis of Hydrodynamics under Dark Fermentation Operating Conditions, Chemical Engineering Journal.

PHOENICS FLAIR – Modelling Aerosol Deposition in Buildings
By Harry Claydon and Mike Malin from CHAM

PHOENICS-FLAIR is equipped with an Eulerian-based multi-phase model for simulating dispersion and deposition of aerosol particles in indoor environments. Typical applications include studying indoor air quality and designing ventilation systems to deal with: human exposure to biological or radiological aerosols in healthcare or laboratory environments; health hazards from industrial aerosols; protective environments and isolated clean rooms; and surface contamination of artworks, electronic equipment, etc. Earlier CHAM Newsletters [1,2] give background on the model theory and 1D duct validation cases. This article focuses on model validation in the 3D case of a clean room.

The aerosol model is validated for the case of steady, isothermal airflow with aerosol transport and deposition of 10µm particles in a laboratory-scale room environment. Surface deposition is computed using the 3-layer deposition model of Chen & Lai (2004) [3], which accounts for the deposition mechanisms of gravity, Brownian and turbulent diffusion. Multiple experimental and numerical results have been reported for this case [4-10]. Particles mainly deposit on the floor, and the model predicts a floor deposition fraction of 69%, which compares favourably with the range of values (60 to 80%) reported by other workers [5, 6] using both Eulerian and Lagrangian CFD models.

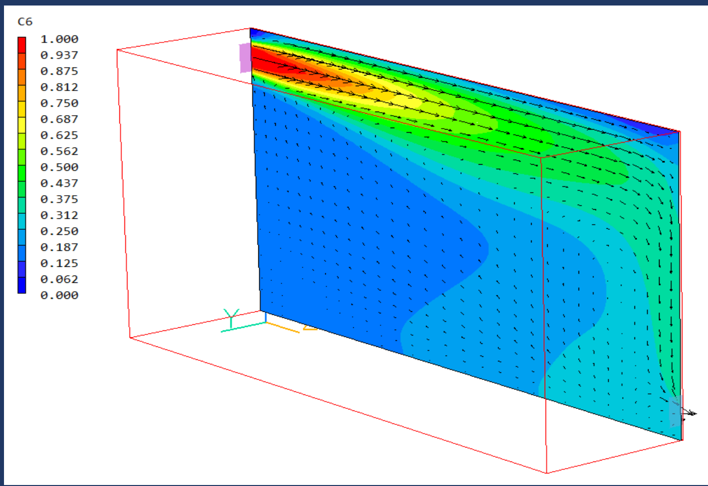


Figure 1: Ventilated Room: Contour plot of particle concentration.

In the experiments, two inlet velocities of 0.225 and 0.45m/s were considered, this simulation uses the lower inflow rate of 0.225m/s and exploits symmetry about the centre plane. Figure 1 shows velocity vectors superimposed on contours of the particle mass fraction normalized by its inlet value ($C6=C/C_{in}$). Vertical profiles of the predicted particle concentration at three different axial stations show a good match with the measurements [4], as seen in Figure 2.

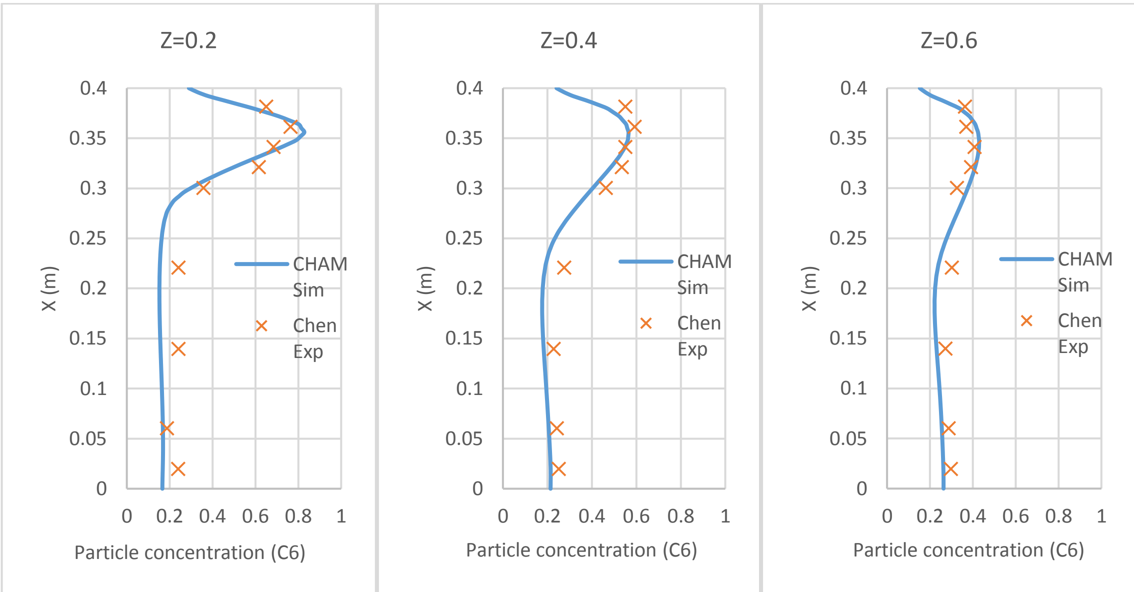


Figure 2: Ventilated Room: Vertical profiles of particle concentration compared with experimental results

In a forthcoming newsletter an article will report on the use of this model in the study of reducing personal exposure to particulate matter from residential Chinese cooking, in conjunction with Tsinghua University. A paper on this has recently been published by Y. Zhao and B. Zhao [11] as part of the International Symposium on Heating, Ventilation and Air Conditioning (2019).

References

1. "PHOENICS Modelling of Indoor Aerosol Transport and Deposition". (2015). *PHOENICS News - CHAM. from* http://www.cham.co.uk/
2. "FLAIR – Drift Flux Model for Aerosol Deposition". (2017). *PHOENICS News - CHAM. from* http://www.cham.co.uk/
3. Chen, F.Z. & Lai, A.C.K, "An Eulerian model for particle deposition under electrostatic and turbulent conditions", J.Aerosol Science, Vol.35, p47-62, (2004).
4. Chen, F.Z., Yu, S.C.M., Lai, A.C.K.,"Modeling particle distribution and deposition in indoor environments with a new drift-flux model", Atmospheric Environment 40, 357–367, (2006).
5. Lai, A.C.K., Chen, F.Z.,"Modeling particle deposition and distribution in a chamber with a two-equation Reynolds-averaged Navier–Stokes model", Aerosol Science 37, 1770–1780, (2006).
6. Lai, A.C.K., Chen, F.Z., "Comparison of a new Eulerian model with a modified Lagrangian approach for particle distribution and deposition indoors", Atmospheric Environment 41, 5249–5256, (2007).
7. Zhao, B., Wu, J. "Particle deposition in indoor environments: Analysis of influencing factors", Journal of Hazardous Materials, Vol. 147, Issues 1–2, page 439-448, (2007).
8. Gao, N.P., Niu, J.L., "Modeling particle dispersion and deposition in indoor environments", Atmospheric Environment 41, 3862-3876, (2007)
9. Zhao, B., C Yang, C., Yang, X.,Liu, S.,"Particle dispersion and deposition in ventilated rooms: testing and evaluation of different Eulerian and Lagrangian models", Building and Environment 43 (4), 388-397, (2008).
10. Xu, G., Wang, J.,"CFD modeling of particle dispersion and deposition coupled with particle dynamical models in a ventilated room", Atmospheric Environment 166, 300-314, (2017).
11. Zhao, Y. and Zhao, B., "Investigations for Reducing Personal Exposure to PM2.5 from Residential Chinese Cooking Based on CFD Simulation". Particulate Matters 2, pp.293, 11th ISHVAC, July 12-15th (2019)

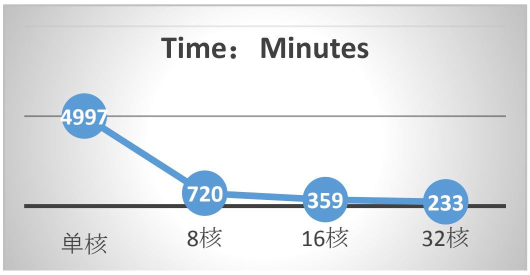
News from CHAM Agents - Shanghai Feiyi

Shanghai Feiyi has carried out speed trials of Parallel PHOENICS (P2019 V1):

1) Conditions: OS:windows 7 SP1 64bit, Processor : intel xeon cpu E5 2698V3, RAM:128GB, PHOENICS 2019 v1 64bit, Grid Number:507*513*110, Area ：6000m*6000m*300m, Sweep:1000
2) Case: Outdoor wind environment
3) Test Result

Core Numbers	Sequential	8-Cores	16-Cores	32-Cores
CPU Time	83 Hours	12 Hours	6 Hours	4 Hours

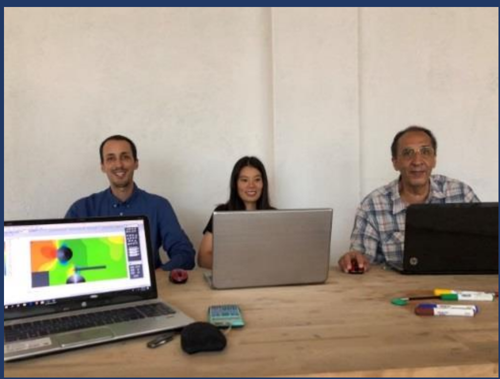
Core Numbers and time

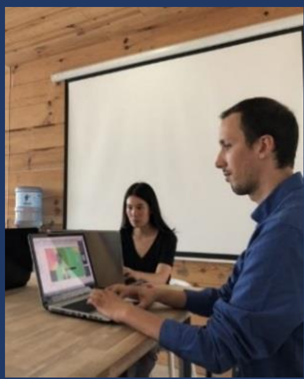


News from CHAM Agents - ArcoFluid

1. ArcoFluid has been a CHAM Agent for many years operating in France and also through its branch in Florida. This year, Mr Zaim Ouazzini, the son of ArcoFluid's founder Dr Jalil Ouazzini, and Mrs Dong Shi Wang, opened a company which will act as a Centre of Excellence for PHOENICS in liaison with Mr Fan of Shanghai-Feiyi, CHAM's Agent in China.The Company, Dali Dongqing Software Technology, is based at 243 San Wen Bi Cun Wei, Dali City, Yunnan. In August Dr Ouazzani visited Dali Dongqing to train staff in the use of PHOENICS and FLAIR.Next April, Jalil will be visiting the Institute of Fluid Mechanics in Beijing to spend a month undertaking a series of PHOENICS-based demonstrations, with his son. These Key Laboratories, as they are called, will be directed to different departments at the Institute.

大 理 冬 青® Dali Dongqing Software Technology

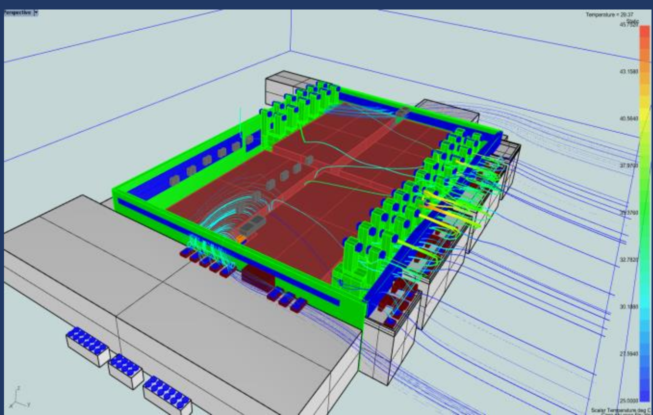




2. In 2019, CHAM and ArcoFluid carried out successful simulations for a large French construction group. Using the FLAIR software both internal and external thermal flows were modelled in a Datacentre located near Paris, France. The numerical simulations have led to significant improvements.

3. Arcofluid and Arcofluid Consultancy are supporting several academic research groups worldwide. An example in the following:

a) The work of an exchange student, Mr Wenjun Liu, between Beijing Institute of Mechanics of Academy of science and Marseille Institute Mediterranean of technology, has led to a conference presentation and a published paper: Wenjun Liu, Paul G. Chen, Jalil Ouazzani, Qiu-sheng Liu. Hydrodynamic Instability of an Evaporating Liquid Layer in a Cylindrical Pool, International Conference Droplets 2019, Sept.16-18,2019, Durham, UK.



Outline of the domain and Temperature coloured Streamlines

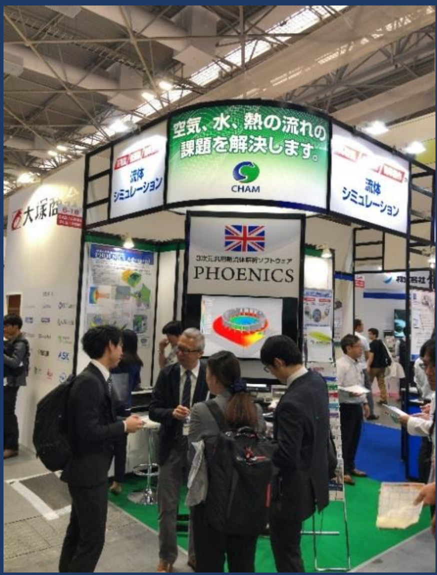
The exchange program is continuing under the physical theme of evaporation/condensation of liquid layers and droplets using the VOF method.

b) University of Clermont Ferrand have been involved with plant growth towards the use in space and in mixing (see article in this newsletter) in the scope of two PHD theses. They have used PHOENICS to simulate these complex physical phenomena. They have published the following two articles:

B. Chezeau, C. Vial, J.-P. Fontaine, A. Danican (2019). Analysis of liquid-to-gas mass transfer, mixing and hydrogen production in dark fermentation process. Chemical Engineering Journal, CEJ-D-19-01137R1, *in progress*.

News from CHAM Japan

CHAM Japan attended the Kansai Design and Manufacturing Solution Exhibition 2019 in Osaka from October 2-4, 2019 (3 days) and also held a User Meeting at their premises in Tokyo on October 18. Seven presentations were made at the User Meeting the titles of which are given below:



1) Hironori Kimura, Ishimoto Architectural & Engineering Firm, Inc. “Application of thermal environment simulation in architectural space”

2) Tomonori Tamura, National Astronomical Observatory of Japan, Advanced Technology Center, “Evaluation of temperature control using heat exchanger in observation room”

3) Prof.Syunichi Sakuragi, Shizuoka Institute of Science and Technology, Faculty of Science and Engineering, “Possibility of floating type high-speed transportation without wings - Development of wingless aerotrain by SIST “

4) Prof.Yasuhiro Shimazaki, Toyohashi University of Technology, Architecture / Urban Systems, “Examples of using CFD by comfortable space design”

5) Yuuji Kubota, Koken Ltd., Advanced Technology Center, Basic Research Laboratory, “Necessary exhaust air volume of local exhaust system determined from exposure concentration of workers”

6) Akira Takenaka, E-tech,Inc., “Study on improvement of hot air velocity distribution at each stage entrance of building material multi-stage drying furnace”

7) Prof.Takaya Akashi, Hosei University, Faculty of Life Sciences, “Visualization of flow in separation and recovery of gallium oxide from waste LED device using spouted bed”

News from CHAM

CHAM would like to take this opportunity to welcome our newest Project Engineer, Harry Claydon, to our London office.

Contact Us

Should you require any further information on any of our offered products or services, please give us a call on +44 (20) 8947 7651. Alternatively, you can email us on sales@CHAM.co.uk

Our website can be viewed at www.CHAM.co.uk and we are on the following social media:

Concentration Heat and Momentum Limited

Bakery House
40 High street
Wimbledon Village
London SW19 5AU, England.