EUROMECH COLLOQUIUM 513:
Dynamics of non-spherical particles in fluid turbulence

BOOK OF ABSTRACTS

Under the auspices of ERCOFTAC SIG 43

Sponsored by:

NTNU – Trondheim
Norwegian University of Science and Technology

Università degli Studi di Udine (Italy)

International Centre for Mechanical Sciences

European Research community on Flow Turbulence and Combustion

EUROPEAN MECHANICS SOCIETY

FINAL PROGRAMME

DAY 1 – Wednesday, April 6, 2011

09:00 – 09:30	Registration

09:30 – 10:00	Welcome

SESSION 1		FIBRE SUSPENSIONS
CHAIR		H. Andersson
TIME	SPEAKER	TITLE
10:00 – 10:40	**C. Aidun**	**Fibre suspension flow inside straight and converging channels**
10:40 – 11:00	**S. Skali-Lami**	**Flocculation of fibres suspension through a planar contraction**
11:00 – 11:20	**M. Putkiranta**	**The effect of channel contraction profile and turbulence on fiber orientation**

11:20 – 11:40	COFFEE BREAK

SESSION 2		PARTICLE DYNAMICS IN FREE AND WALL-BOUNDED FLOWS
CHAIR		F. Lundell
TIME	SPEAKER	TITLE
11:40 – 12:00	**F. Zhang**	**Disturbance growth during sedimentation in dilute fibre suspension**
12:00 – 12:20	**E. Variano**	**Rotation dynamics of ideal nonspherical particles and extension to field measurements**
12:20 – 12:40	**R. van Hout**	**Experimental investigation of non-spherical pollen grain settling in near homogeneous isotropic turbulence**
12:40 – 13:00	**A. Seeliger**	**Experiments to assess the transport of fibrous insulation debris**

13:00 – 14:00	LUNCH

SESSION 3		PARTICLE DYNAMICS IN FREE AND WALL-BOUNDED FLOWS
CHAIR		J. Hamalainen
TIME	SPEAKER	TITLE
14:00 – 14:20	**F. Marra**	**Numerical modeling of char particles segregation in entrained-flow slagging gasifiers**
14:20 – 14:40	**R. Powell**	**Transport effects in fibrous suspensions**
14:40 – 15:00	**L. Tognotti**	**Non-spherical particle sub-models in comprehensive modelling of combustion systems**
15:00 – 15:20	**K. Volkov**	**Dynamics of non-spherical compound metal particle in non-uniform flow field**

15:20 – 15:50	COFFEE BREAK

SESSION 4		PARTICLE DYNAMICS IN FREE AND WALL-BOUNDED FLOWS
CHAIR		C. Aidun
TIME	SPEAKER	TITLE
15:50 – 16:10	**C. Marchioli**	**Orientation, distribution and deposition of inertial fibers in turbulent channel flow**
16:10 – 16:30	**M. Kvick**	**Analysis of particle streaks**
16:30 – 16:50	**K Hakansson**	**Streak formation and fibre orientation in near wall turbulent fibre suspension flow**
16:50 – 17:10	**L. Zhao**	**Two-way coupled simulations of ellipsoidal particles suspended in a turbulent channel flow**

Fibre suspension flow inside straight and converging channels

Cyrus K Aidun
G.W. Woodruff School of Mechanical Engineering,
Georgia Institute of Technology, Atlanta, USA

The behavior of rigid and deformable fibers suspended in turbulent flow in a straight channel and a planar contraction will be discussed. The effect of volume fraction and the channel Reynolds number on velocity profile inside the channel will be presented based on Pulsed Ultrasonic Doppler Velocimeter.

The impact of turbulence on orientation anisotropy in a converging channel based on measurement of orientation at different streamwise positions with clearly defined turbulent conditions at the inlet and turbulent flow variations along the contraction will be considered next. First the case of dilute fiber suspension where $nL^3<1$ will be discussed. In this case, fibre–fibre interactions and the effect of the fibres on the flow rheology become negligible. Nearly homogeneous isotropic grid turbulent flow is introduced at the channel inlet and its variation in the contraction is measured. Since the influence of turbulence on orientation anisotropy can be expressed by an orientational diffusion coefficient, the factors affecting this coefficient will be discussed. The effect of inlet flow characteristics in contrast to turbulence produced in the contraction will be presented. Specifically, it will be shown that turbulent intensity decays exponentially with contraction ratio, C, where C is defined as the ratio of the local height and the inlet height . At $C > 4$, turbulent intensity becomes very small but finite due to small production of turbulent energy. However, in this region (i.e., $C > 4$) the effect of turbulence on fibre orientation becomes negligible where the effect of mean velocity gradient on fibre orientation becomes dominant. To effectively represent the competing roles of turbulence and mean velocity gradient on fibre orientation distribution in the contraction, the results will be presented based on the rotational Peclet number.

The effect of fibre deformation on rheology of dilute to concentrated fibre suspension flow will be discussed based on direct computational simulation of fiber suspension in shear flow.

References

Wu, J. and Aidun, C.K., "A numerical study of the effect of fiber stiffness on the rheology of sheared flexible fiber suspensions, *J. Fluid Mech.*, vol. 662, pp. 123–133, 2010.

Aidun, C.K., and Parsheh, M., "Counter-rotating core in swirling turbulent flow inside a pipe," *Phys Fluids*, 19, 061704, 2007.

Parsheh M., Brown, M., and Aidun, C.K., "Fiber Orientation in a Planar Contraction: Shape Effect" *Int. J. Multiphase Flow*, 32, 1354-1369, 2006.

Brown, M., Parsheh, M., and Aidun, C.K., "Turbulent Flow in a Converging Channel: Effect of Contraction and Return to Isotropy," *J. Fluid Mech.*, 560, 437-448, 2006.

Xu, Hanjiang and Aidun, C.K., "Characteristics of Fiber Suspension Flow in a Rectangular Channel" *Int. J. Multiphase Flow*, 3 1/3, pp. 318-336, 2005.

Flocculation of fibres suspension through a planar contraction

S. SKALI LAMI

Nancy-Université

LEMTA : Laboratoire d'Energétique et de Mécanique Théorique et Appliquée

2 Av. de la Forêt de Haye - BP 850 . 54504 Vandoeuvre les Nancy. France

Abstract

The flow of fibre suspensions are of interest in the pulp and paper industry. During the paper forming process, at the start of the paper machine, a dilute fibre suspension flows through a headbox. The fluid passes through flow distributors and turbulence generators designed to produce high turbulence intensity, the utility of this part is to dislocate the fibre network into the flocs. The flow subsequently passes through a planar contraction that accelerates the fluid to a high speed and creates a thin planar jet. The flocs formed after the turbulence generator then undergo an acceleration during contraction. In this paper we analyse the elongation of flocs with simplified theoretical approach.

The effect of channel contraction profile and turbulence on fiber orientation

Maria Putkiranta[1], Hannu Eloranta[1], Tero Pärssinen[2] and Pentti Saarenrinne[1]

[1]Tampere University of Technology, Department of Energy and Process Engineering
P.O. Box 589, Tampere, FI-33101, Finland
[2]Metso Paper Inc., P.O. Box 587, FI-40101 Jyväskylä, Finland

Corresponding author: maria.putkiranta@tut.fi

Fiber orientation distribution is known to determine paper strength properties and its dimensional stability . Fibers get aligned in the streamwise direction in the headbox slice due to an accelerating base flow. The final fiber orientation distribution is also affected by other factors, such as turbulence level in the headbox, the slicebar design and the jet-to-wire ratio.

This paper presents the development of fiber orientation distribution in streamwise accelerating flow fields in three different channel profiles. All measurements are repeated with four fiber types (long and short rayon, euca and pine) to see the effect of fiber properties. The fiber orientation in the flow is measured by taking images the fiber suspension and using image analysis to determine the local fiber orientation. The results show the differences in the magnitude of orientation between different fibers. There are also significant differences in orientation produced by the different channel profiles even though the contraction ratio in all channel profiles is the same. Also PIV measurements are done in all the profiles and the results are reflected to fiber orientation results.

After that the effect of turbulence on fiber orientation is studied in a channel profile with flat walls, which was found to be the profile producing the highest fiber orientation anisotropy. The turbulence in the channel is varied by changing location and the size of holes in a turbulence generator. Using smaller holes and locating the turbulence generator closer the contraction increases the turbulence level and randomizes the fiber orientation distribution. Results for euca fibers are shown in Fig. 1 for higher turbulence level (8mm holes) and lower turbulence level (12 mm holes) at two different flow rates. To investigate the differences in turbulence intensity, PIV measurements are done in all tested cases.

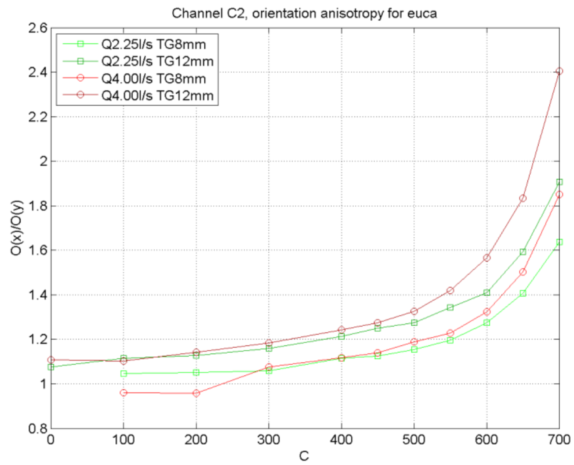


Figure . Effect of turbulence generator on fiber orientation anisotropy in channel C2.

Disturbance growth during sedimentation in dilute fibre suspension

Feng Zhang[1], Anders Dahlkild[1] & Fredrik Lundell[2]
[1] *Linné FLOW Centre* [2] *Wallenberg Wood Science Centre*
KTH Mechanics, Royal Institute of Technology, Stockholm S-100 44, Sweden

It is well known that dilute suspensions of fibres settling under gravity at low Reynolds number (*Re*) are unstable to density fluctuations. Koch and Shaqfeh (1989) predicted this instability in the absence of inertia and diffusive effects. The mechanism identified by them relies on the coupling between the orientation of the settling nonspherical particles and the flow field induced by the density fluctuations. Using a linear stability analysis in an unbounded homogeneous suspension they found that the normal mode density fluctuations with the maximum growth rate are those of infinite horizontal wavelength. In a bounded system, the container size limits the size of the largest wavelength. However, they didn't believe that such long wavelength fluctuations will in fact develop. The growth of a single streamer spanning the width of the box was reported in the simulations [Saintillan et al. (2005), Tornberg et al. (2006)] at short times in periodic suspensions, confirming that the longest wavelength is the most unstable in such systems. However, according to experiments of Herzhaft et al. (1998) and Metzger et al. (2007), wavelengths equal to the container dimension were not observed at early times.
Here, the early stage of sedimentation is studied using a linear stability analysis of Fokker-Planck equation with self-diffusion coupled to the Navier-Stokes equation with inertia, it is shown that inertia and self-diffusion damp long wavelengths and short wavelengths, respectively, leading to wavenumber selection. For small, but finite *Re* of the fluid bulk motion, the most unstable wavenumber is not zero any more and increases with *Re*. For wavenumber zero, the growth rate is zero instead of attaining its maximum as for zero *Re*. However, for small *Re* the influence of inertial effects on the dispersion relation is restricted to small wavenumbers. The velocity fluctuations of particles in dilute suspensions can lead to randomly fluctuating motions, which have a long time behavior characteristic of a diffusion process known as hydrodynamic self-diffusion. The effect of including self-diffusion is to damp the density fluctuations of the linear stability analysis only in the range of large wavenumbers. We also investigate the influence of the initial conditions. It is shown that just in the early development of the perturbation, the growth rates might differ from exponential growth of a single linear mode. Except this, the different initial conditions only have slight effect on the relationship between wavenumbers and growth rates.
Forthcoming work will focus on long times simulations of the non-linear development in bounded systems to estimate the effect of rotational diffusion and walls.

References

D. L. Koch and E. S. G. Shaqfeh, J. Fluid Mech. 209, 521(1989).
D. Saintillan, E. Darve, and E. S. G. Shaqfeh, Phys. Fluids 17, 033301(2005).
A.-K. Tornberg and K. Gustavsson, J. Comput. Phys. 215, 172 (2006).
B. Herzhaft and ÃĽ. Guazzelli, J. Fluid Mech. 384, 133 (1999).
B. Metzger, J. E. Butler, and ÃĽ. Guazzelli, J. Fluid Mech. 575,307 (2007).

Rotation dynamics of ideal nonspherical particles and extension to field measurements

Evan A. Variano[1,2], Ian Tse[1], Gabriele Bellani[1]

[1]KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden
[2]Civil and Environmental Engineering, University of California Berkeley, Berkeley (CA), USA
bellani@mech.kth.se, fredrik@mech.kth.se
Keywords: Isotropic turbulence, PIV

Laboratory measurements using manufactured refractive-index matched particles give the full rotation vector for spherical and non-spherical particles. We compare the PDF of angular velocity across particle shape and timescale. To extend this work to field studies in the environment,we have constructed a rugged, field-deployable camera that measures Lagrangian particle paths. To demonstrate the utility of this we present the velocity statistics of living plankton in turbulent flow.

Experimental investigation of non-spherical pollen grain settling in near homogeneous isotropic turbulence

René van Hout*, Lilach Sabban
Faculty of Mechanical Engineering
Technion - Israel Institute of Technology
Haifa, 32000, Israel
*rene@technion.ac.il

In models of atmospheric pollen dispersal, pollen grains are generally modelled as small spheres and any morphological features are thought to be irrelevant to their dispersal characteristics. However, most pollen grains have striking morphological features such as spikes and air sacs that may affect dispersal. Here, we present results on non-spherical pollen grains settling in near Homogeneous, Isotropic Turbulence (HIT). The turbulence is generated in a 40 cm^3 transparent turbulence chamber by 8 woofers mounted on the corners of the chamber [1,2]. Near HIT in the center of the chamber was validated for two woofer amplifications using stereoscopic Particle Image Velocimetry (PIV) with a field of view of ~45 x 45 mm^2. Ratios of rms values of fluctuating velocity components were close to one, indicating isotropy in the center of the chamber. Longitudinal and transverse Turbulent Kinetic Energy (TKE) spectra calculated from the PIV data sets, collapsed. In order to provide an estimate for the Kolmogorov scales and particle Stokes numbers, TKE dissipation rates were estimated using three different methods. Resulting Kolmogorov length and time scales were ~180 µm and ~2ms with Taylor scale Reynolds numbers of about 200.

Corn (*Zea Maize*, 80µm), pine (*Pinus Taeda*, 60µm, Fig. 1), ragweed (*Ambrosia,* 20µm) pollen as well as polystyrene spheres with similar diameter and density as corn, were released into the chamber. Settling trajectories were measured using high-speed (up to 1 kHz), inline digital holographic cinematography using a lensless CMOS camera (Photron Ultima APX, 1k x 1k, 17x17 μm^2 pixels). The volume of interest was 17x17x40 mm^3 located in the center of the chamber. Particle Stokes numbers ranged from 0.1 to 10; the lowest value for ragweed pollen. Holographic movies of particle settling were captured in still air and in near HIT. Between 100 to 700 particles were tracked for each particle type with an average tracking distance of about 40 frames. Still air settling velocities and deduced pollen densities compared well to values found in the literature. In all cases, settling velocities in near-homogeneous, isotropic turbulence are enhanced over their still air settling values; the enhancement is highest for the lowest Taylor Reynolds number. Especially large settling velocity enhancements were obtained for pine that exhibits the most striking morphological features in the form of air sacs (Fig. 1). The settling enhancement of corn pollen and polystyrene spheres were similar. The data is currently further processed to obtain diffusion coefficients and results will be presented. In addition higher resolution (~ 3-5µm/pixel) high speed holographic cinematography is underway in order to resolve the possible changing orientation of non-spherical pollen grains while settling in near HIT.

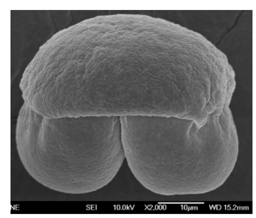


Fig. 1. Pine pollen

References

[1] Hwang W., Eaton J. K. (2004) Creating homogeneous and isotropic turbulence without a mean flow. *Exp. Fluids*, Vol. 36, 444-454.

[2] Sabban L., van Hout R. (2010) Measurement of pollen settling characteristics in near homogeneous isotropic turbulence. *Proceedings of the 7th International Conference on Multiphase Flow*, ICMF 2010, Tampa, FL USA, May 30-June 4.

Experiments to assess the transport of fibrous insulation debris

A. Seeliger[1], G. M. Cartland Glover[2], E. Krepper[2] S. Renger[1], W. Kästner[1] and H. Kryk[2]

[1] Hochschule Zittau/Görlitz, Institute of Process Technology, Process Automation and Measuring Technology, P.O. Box 1455, D-02754, Zittau, Germany

[2] Helmholtz-Zentrum Dresden-Rossendorf, Institute of Safety Research, P.O. Box 510119, D-01314, Dresden, Germany

ASeeliger@hs-zigr.de, g.glover@fzd.de, e.krepper@fzd.de, srenger@hs-zigr.de

Abstract

Loss of coolant accidents (LOCA) in the primary cooling circuit of a nuclear reactor may result in damage to insulation materials that are located near to the leak. The insulation materials released may compromise the operation of the emergency core cooling system (ECCS). Insulation material in the form of mineral wool fibre agglomerates (MWFA) maybe transported to the containment sump strainers mounted at the inlet of the emergency cooling pumps, where the insulation fibres may block or penetrate the strainers. In addition to the impact of MWFA on the pressure drop across the strainers, corrosion products formed over time may also accumulate in the fibre cakes on the strainers, which can lead to a significant increase in the strainer pressure drop and result in cavitation in the ECCS. Thus, knowledge of transport characteristics of the damaged insulation materials in various scenarios is required to help plan for the long-term operability of nuclear reactors, which undergo LOCA.

An experimental and theoretical study performed by the Helmholtz-Zentrum Dresden-Rossendorf and the Hochschule Zittau/Görlitz[1] is investigating the phenomena that maybe observed in the containment vessel during a LOCA. The study entails the generation of fibre agglomerates, the determination of their transport properties in single and multi-effect experiments and the long-term effect that corrosion of the containment internals by the coolant has on the strainer pressure drop.

The focus of this presentation is on the experiments performed that characterize the horizontal transport of MWFA, whereas the corresponding CFD simulations are described in an accompanying contribution (see abstract of Cartland Glover et al.). The experiments were performed a racetrack type channel that provided a near uniform horizontal flow. The channel is 0.1 wide by 1.2 m high with a straight length of 5 m and two bends of 0.5 m. The measurement techniques include particle imaging (both wide-angle and macro lens), concurrent particle image velocimetry, ultra-velocimetry, laser detection sensors to sense the presence of absence of MWFA and pertinent measurements of the MWFA concentration and quiescent settling characteristics. The transport of the MWFA was observed at velocities of 0.1 and 0.25 m s^{-1} to verify numerical model behaviour in and just beyond expected velocities in the containment sump of a nuclear reactor.

Keywords: Mineral Wool Fibre Agglomerates, Loss of Coolant Accidents, Containment Sump, Particle Imaging, Particle Image Velocimetry,

---

[1] The reported investigations are funded by the German Federal Ministry of Economics and Technology (BMWi) under Contract No. 1501360 and 1501363.

Numerical Modeling of Char Particles Segregation in Entrained-Flow Slagging Gasifiers

Fiorenzo Ambrosino[a], Andrea Aprovitola[b], Francesco Saverio Marra[b], Fabio Montagnaro[c], Piero Salatino[d]

[a]*ENEA - Agenzia Nazionale per le Nuove Tecnologie, l'Energia e lo Sviluppo Economico Sostenibile, CR Portici, Piazzale Enrico Fermi 1, 80055 Portici (Italy)*

[b]*Istituto di Ricerche sulla Combustione, Consiglio Nazionale delle Ricerche, Via Diocleziano 328, 80124 Napoli (Italy)*

[c]*Dipartimento di Chimica, Università degli Studi di Napoli Federico II, Complesso Universitario del Monte di Sant'Angelo, 80126 Napoli (Italy)*

[d]*Dipartimento di Ingegneria Chimica, Università degli Studi di Napoli Federico II, Piazzale Vincenzo Tecchio 80, 80125 Napoli (Italy)*

This work illustrates how different numerical modeling approaches can be complementarily adopted to gain the knowledge necessary to explain the observed segregation/entrapment of char particles on/into the molten slag layer in full-scale entrained-flow slagging coal gasifiers *[1–6]*. A multi-level approach is being developed: RANS-based simulations of the full-scale geometry with coal particle injection and tracking aims to obtain the general features of the flow field and the particle trajectories, allowing to estimate the effect of swirl and turbulence on the char particle deposition rate *[7]*.

These results are adopted in two different subsequent models: a reduced plug-flow based model of the full-scale geometry and a more detailed model of a particle-laden channel flow confined by a slag layer. This last model, based on the solution of the filtered Navier-Stokes equations, is solved with the adoption of the OpenFOAM toolkit *[8]*. An LES approach for the turbulent Eulerian gas phase is applied *[9]*. The equations of particles motion are solved via a Lagrangian particle tracking algorithm with the TrackToFace method *[10]*. Simulations have been performed involving a number of particles from 10^5 to 10^6, which is considered sufficient to obtain a clear picture of the multiphase flow behaviour responsible for char deposition phenomena *[11,12]*. This multi-level approach allows to assess the char particle deposition rates and the nature of char/slag interaction (segregation/entrapment) that are likely to occur in full-scale slagging gasifiers.

Literature Cited:

[1] Wu, T., Gong, M., Lester, E., Wang, F., Zhou, Z., Yu, Z.: *Fuel*, **86**:972–82 (2007).
[2] Li, S., Whitty, K.J.: *Energy Fuels*, **23**:1998–2005 (2009).
[3] Xu, S., Zhou, Z., Gao, X., Yu, G., Gong, X.: *Fuel Process. Technol.*, **90**:1062–70 (2009).
[4] Brachi, P., Montagnaro, F., Salatino, P.: *Proc. 33rd Meeting Italian Section Combustion Institute*, ptse2010.P1.11, Ischia, Italy (2010).
[5] Montagnaro, F., Salatino, P.: *Combust. Flame*, **157**:874–83 (2010).
[6] Zhao, X., Zeng, C., Mao, Y., Li, W., Peng, Y., Wang, T., Eiteneer, B., Zamansky, V., Fletcher, T.: *Energy Fuels*, **24**:91–4 (2010).
[7] Sommerfeld, M., Qiu, H.-H.: *Int. J. Multiphas. Flow*, **19**:1093–127 (1993).
[8] Weller, H.G., Tabor, G., Jasak, H., Fureby, C.: *Comput. Phys.*, **12**:620–31 (1998).
[9] Yoshizawa, A., Horiuti, K.: *J. Phys. Soc. Jpn.*, **54**:2834–9 (1985).
[10] Macpherson, G.B., Nordin, N., Weller, H.G.: *Commun. Numer. Meth. En.*, **25**:263–73 (2009).
[11] Marchioli, C., Soldati, A., Kuerten, J., Arcen, B., Tanière, A., Goldensoph, G., Squires, K., Cargnelutti, M., Portela, L.: *Int. J. Multiphas. Flow*, **34**:879–93 (2008).
[12] Marchioli, C., Salvetti, M., Soldati, A.: *Acta Mech.*, **201**:277–96 (2008).

Transport Effects in Fibrous Suspensions

E. Tozzi, D. Lavenson, M. McCarthy and R. Powell
Department of Chemical Engineering and Materials Science
University of California Davis

Magnetic resonance imaging (MRI) is used to study the dynamics of fibrous suspensions, which are important in many industrial processes especially pulp and paper processing and cellulosic biofuel production. MRI allows real, opaque systems to be studied. This talk will briefly review the MRI methodology for concentration, velocity profile and rheological measurements. Results for a variety of fibrous suspensions will be presented. Processing of fibrous biomass requires adequate characterization of the fluid mechanics and rheology of fiber suspensions. Due to the large size of biomass particles, fast settling, entanglements and migration occur. Direct imaging of velocity profiles provides a way of characterizing flow in the presence of such non-idealities. We used magnetic resonance imaging to measure velocity profiles of cellulosic fiber suspensions flowing in a horizontal tube. Pressure drop was also recorded. We observed a strong influence of fiber length, concentrations and flow rates on velocity profiles and pressure drops. Various types of flow were observed including immobile layers at the bottom of the pipe, mobile networks that translate as a plug and also velocity profiles that suggest smooth concentration gradients along the vertical direction. The concentration effects were best described by the use of a crowding number, with large changes in pressure and velocity profiles occurring in a narrow range of crowding numbers. Qualitative differences between the behavior of the long fibers and the short and medium fibers demonstrate a strong effect of fiber aspect ratio on rheology.

Euromech Colloquium 513, Dynamics of
non-spherical particles in fluid turbulence
6-8 April 2011, Udine, Italy.

Non-spherical particle sub-models in comprehensive modelling of combustion systems

E. Biagini, C. Galletti, L. Tognotti
Dipartimento di Ingegneria Chimica, Università di Pisa

Non-spherical particles are largely encountered in combustion flows, especially for the cases of biomass particles (e.g., coal-biomass cofiring in utility boilers). For instance pulverized straw particles are highly non-spherical and mainly flake-like, whereas pulverised wood is rod-like. In these cases the assumption of spherical particles in combustion models may lead to discrepant results under different aspects. The introduction of a shape factor enhances the surface area compared to that of a sphere, thus affecting not only the particle motion but also heating and surface reactions. The present work discusses some modelling critical issues related to the non-sphericity of solid fuel particles.

Firstly an accurate knowledge of size and shape distribution is needed. Existing data are scarce and not always quantitative. Particle size and shape distributions based on detailed characterization procedures should be applied to parent fuels and high heating rate chars to provide fundamental parameters for studying primary devolatilization and char oxidation, respectively (Biagini et al. 2009). Information on structural parameters (e.g., swelling, shrinking, fragmentation) can be obtained for non-spherical particles, also considering the anisotropic nature of biomass fuels (Kumar et al. 2006).

Secondly, commercial CFD codes allows generally to include the effect of non-sphericity on the particle motion by including a modification in the steady state drag coefficient (e.g. Haider and Levenspiel). The turbulence modulation may be important as pulverized biomass particles are generally bigger and more elongated than coal particles, thus they are likely to increase turbulence. So it is important to determine the effect of turbulence modulation on the efficiency of combustion. However there is a lack of validation of existing turbulence modulation models.

A further aspect is related to the particle heating. Generally the available models in commercial CFD codes are based on the spherical assumption, so that it would be necessary to develop some efficiency factor based on particle sphericity to correct the convective and radiative heat transfer (Mandø et al., 2010).

Due to different surface area available, elongated particles heat more rapidly than equal volume spherical particles. Biomass volatiles are thus released earlier with crucial implications on flame stability, oxygen consumption and eventually pollutant formation (e.g., NOx). Also the heterogeneous reactions with oxygen are generally based on the external surface area, so the proper description of the non-spherical particles is recommended to simulate the subsequent char oxidation step. Size reduction and fragmentation during this step need a specific approach to predict differences with respect to the case of spherical particles.

Finally the interactions between motion and combustion during all steps should be described to achieve a better understanding on biomass combustion.

Biagini E, Simone M, Tognotti L. Characterization of high heating rate chars of biomass fuels. Proceedings of the Combustion Institute 32 (2009) 2043–2050

Haider A, Levenspiel O. Drag coefficient and terminal velocity of spherical and nonspherical particles. Powder Technol 58 (1989) 63–70.

Lu H, Ip E, Scott J, Foster P, Vickers M, Baxter LL. Effects of particle shape and size on devolatilization of biomass particle. Fuel 89 (2010) 1156–1168

Mandø M, Rosendahl L, Yin C, Sørensen H. Pulverized straw combustion in a low-NOx multifuel burner: Modeling the transition from coal to straw. Fuel 89 (2010) 3051-3062.

Kumar RR, Kolar AK, Leckner B. Shrinkage characteristics of Casuarina wood during devolatilization in a fluidized bed combustor. Biomass and Bioenergy 30 (2006) 153–165

Dynamics of Non-Spherical Compound Metal Particle in Non-Uniform Flow Field

Konstantin Volkov[1], Vladislav Emelyanov[2], Irina Kurova[2]

[1]Centre for Fire and Explosion Studies, Faculty of Engineering, Kingston University, Friars Avenue, Roehampton Vale, SW15 3DW, London, UK, e-mail: k.volkov@kingston.ac.uk

[2]Department of Gas and Plasma Dynamics, Faculty of Physical and Mechanical Sciences, Baltic State Technical University, 1-ya Krasnoarmeyskaya ul., 1, 190005, Saint Petersburg, Russia

Aluminized composite propellants used in solid rocket motors (SRM) contain a lot of metal (usually aluminium) particles because high combustion energy is generated and propulsion efficiency increases by burning metal particles [1]. To accurately design and predict the performance of SRMs with an aluminized propellant, the combustion behaviour of the aluminium particles and their dynamics in the non-uniform internal flow field is required. While many studies have centred on measuring and modelling burn rate characteristics of propellants, less effort has been focused on aerodynamics, heat transfer and combustion characteristics of agglomerates consisting of metal and its oxide and moving in non-uniform flow occurring in SRMs.

The major product of aluminium combustion is liquid aluminium oxide, which is formed from the condensation of aluminium sub-oxides. A fraction of the oxide diffuses back and deposits on the particle surface and is termed as the oxide cap (lobe). The oxide cap tends to accumulate on the lower end of the particle. The accumulation of the oxide on the particle surface and the porosity of the oxide cap result in a final oxide cap size of the order of the initial particle size. The other fraction of the oxide is transported outwards and is termed as the oxide smoke. The oxide cap results in fragmentation and jetting. The oxide cap effect on the burning time depends on the initial size of the particle [2].

The oxide cap acts as a dead weight which reduces the vaporization surface, leading to non-symmetrical combustion and a possible droplet spin. The oxide lobe affects the vaporization process. The outward flow decreases and becomes non-uniform. Concurrently, the inward diffusive flow from the vapour-phase flame zone increases at the surface region overlapped with oxide. This coupling of the diffusion rate with the outward flow leads to a possible growing of the oxide cap.

The particles are not stationary during combustion, and typically decelerate during combustion. The effect of forced convection on the burning rate and ignition delay is unknown, but remains of significant concern. Several questions remain about unsteady processes, such as spinning and fragmentation of particles. One important phenomenon is the formation of an aluminium oxide cap moving on the particle surface during combustion and movement of particle. It causes particle asymmetry and obstructs part of the aluminium surface, leading to spinning.

The development of robust models of aluminium particle dynamics and morphology is essential in the development of advanced propulsion systems. The study focuses on the numerical analysis of non-uniform flow field around moving aluminium particle with oxide cap inside combustion chamber of SRM. The mathematical model of two-phase flow around a single aluminium droplet with oxide cap has been developed and validated against experimental data. The model solves the continuity, momentum, energy and species continuity equations simultaneously to obtain the species and temperature profiles and the burn time. These equations allow calculating forces and angular moment acting on the particle. Geometry of the aluminium droplet and its oxide cap changes as mass is removed from the aluminium particle and oxide cap mass increases during aluminium combustion. It has been assumed that the oxide deposits uniformly on the particle surface and migrates to the downstream side to coalesce into an oxide cap.

References

1. Volkov K.N., Emelaynov V.N. (2008) Gas-particle flows. Moscow, Publishing House of Physical and Mathematical Literature.
2. Volkov K.N. (2010) Combustion of single aluminium droplet in two-phase flow. In: Heterogeneous Combustion. Edited by G.I. Stoev. USA, Nova Science.
3. Volkov K. (2010) Internal turbulent two-phase flows formed by wall injection of fluid and particles. Proceedings of the 5th European Conference on Computational Fluid Dynamics (ECCOMAS CFD 2010), 14-17 June 2010, Lisbon, Portugal, Paper No. 01649.

Euromech Colloquium 513, Dynamics of
non-spherical particles in fluid turbulence
6-8 April 2011, Udine, Italy.

Orientation, distribution and deposition of elongated, inertial fibers in turbulent channel flow

Cristian Marchioli and Alfredo Soldati *
Dipartimento di Energetica e Macchine, Università di Udine, Udine, 33100, Italy

Suspensions of elongated rigid fibers in turbulent flows are commonly encountered in applications of engineering interest. Examples include pulp production and paper making, where controlling the rheological behavior and the orientation distribution of fibers is crucial to optimize production operations. In addition, recent experiments [1] demonstrated that elongated fibers may represent a feasible alternative to flexible polymers for reducing pressure drops in fluid transport systems. Despite this practical importance, very few phenomenological studies aimed at a deeper understanding of the physics of turbulent fiber transport are available [2, 3]: as a result, current knowledge of the mechanisms govering fibers-turbulence interaction is not satisfactory and requires extensive investigation. Our study is intended to complete the abovementioned studies adding new data in the parameter space (λ, τ_p and Re_τ), where λ is fiber elongation, τ_p is fiber response time, and Re_τ is the shear Reynolds number. New statistics will be presented to fully characterize fiber behavior, and highlighting the circumstances in which fibers behavior significantly deviates from that of spherical particles. To this aim, fiber dispersion in a turbulent channel flow at $Re_\tau = 150$ is first investigated using Direct Numerical Simulation and Lagrangian tracking under the one-way coupling approach. Fibers are treated as prolate ellipsoidal particles which move according to inertia and to hydrodynamic drag and rotate according to hydrodynamic torques. Results are presented for $\lambda = 1, 3, 10, 50$, and $\tau_p^+ = 1, 5, 30, 100$. Superscript + is used here to represent dimensionless variables expressed in wall units.

The orientational behavior of fibers is examined together with their preferential distribution, near-wall accumulation, and wall deposition: all these phenomena are interpreted in connection with turbulence dynamics near the wall. Results confirm that, in the vicinity of the wall, fibers tend to align with the mean streamwise flow direction. However, this aligned configuration is unstable, particularly for higher inertia of the fiber, and can be maintained for rather short times before fibers are set into rotation in the vertical plane. The situation complicates in the spanwise and wall-normal flow directions, where fiber inertia and elongation destabilize near-wall alignment in a non-trivial fashion. Fiber orientational behavior and fiber translational behavior are also observed to influence the process of fiber accumulation at the wall: compared to the case of spherical particles, the elongation has little or no effect on segregation; yet it does affect the wallward drift velocity of the fibers in such a way that longer fibers tend to deposit at higher rates. No preferential orientation and no significant segregation is observed in the channel centerline.

In the final paper, one-way coupling results will be complemented by and compared to new results coming from simulations that include two-way coupling effects. This step of the work aims at exploring from a physical viewpoint the mechanism of fiber-induced turbulent drag reduction and will assist in highlighting differences between coupled and uncoupled predictions of fiber preferential orientation and distribution.

References

[1] Paschkewitz J.S., Dmitropoulos, C.D., Hou, Y.X., Somandepalli, V.S.R., Mungal, M.G., Shaqfeh E.S.G., and Moin, P. (2005) Phys. Fluids, 17, 085101.

[2] Mortensen P.H., Andersson H.I., Gillissen J.J.J., and Boersma B.J. (2008) Phys. Fluids, 20, 093302.

[3] Zhang H., Ahmadi G., Fan F.G., and McLaughlin J.B. (2001) Int. J. Multiphase Flow, 27, 971-1009.

*Corresponding Author: Email: soldati@uniud.it; Phone: +39 0432 558020; Fax: +39 0432 558027

Analysis of particle streaks

M. Kvick*, K. Håkansson*, F. Lundell*†, L. Prahl Wittberg*†, and L. D. Söderberg*‡

* Wallenberg Wood Science Center, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

† Linné FLOW Centre, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

‡ Innventia AB, Box 5604, SE-114 86 Stockholm, Sweden

kvick@mech.kth.se, karlh@mech.kth.se, fredrik@mech.kth.se

Keywords: Streaks, Structures, Image analysis

For particles in turbulent wall bounded flow, it has been found that the particles tend to agglomerate in streamwise streaks close to the wall for certain flow situations. The analysis and quantification of these streaks are not straightforward and methods need to be developed. Both the width of these streaks, and a measure of the streakiness are wanted.

In this study, a method based on correlation analysis is suggested. The method uses the location of each particle in an image, and assigns each particle the same width in the spanwise direction, in order to get high enough energy in the signal. Thereafter, each individual image is summed in the streamwise direction, and an intensity signal is obtained. Figure 1(a) shows two signals for images from two different cases. When autocorrelating each signal, with the mean subtracted, a measure the mean streak-width is found as displacement at which the minimum correlation occurs. The value of the minimum correlation is a measure of how coherent the structures in the flow are.

This in turn should also provide a quantification of the streakiness. However, this is not the case. The two signals in figure 1(a), results in the same value of the minimum correlation. However, when investigating the images by eye, it is evident that the structures in the two cases are of different strength. To achieve a way to quantify the streakiness, several methods have been evaluated. One of the more promising, being a Voronoi analysis, where the *rms* of the summed Voronoi diagram is evaluated. Common for all methods is that the effect of concentration on the correlation is found to be severe. In short, the number of particles in each image effects the minimum correlation. This is also verified in experiments from the same flow with different (very dilute) concentrations.

The concentration effect can also be studied by random removal of particles in the post-processing of images. This way, one can determine the minimum value of the correlation at different concentration levels artificially, see figure 1(b). The minimum correlation increases as particles are removed (the number of particles in each image decreases) and the decrease is seen to be logarithmic. Streakiness measures from experiments at different concentrations can now be compared by interpolation.

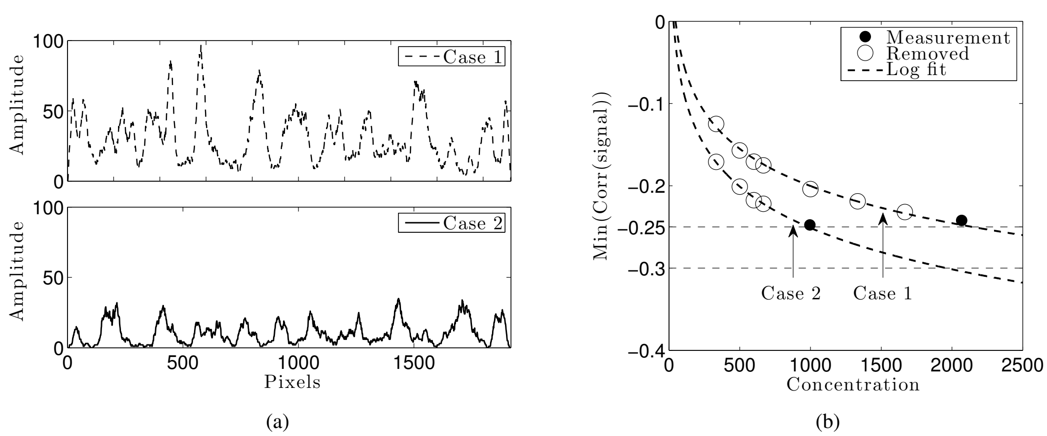


Figure 1: (a) Intensity signal for two images, summed in the direction of the streaks. (b) Correlation minima, where particles have been removed to reach different concentrations.

Streak formation and fibre orientation in near wall turbulent fibre suspension flow

K. Håkansson*, M. Kvick*, F. Lundell†, L. Prahl Wittberg† and L. D. Söderberg*

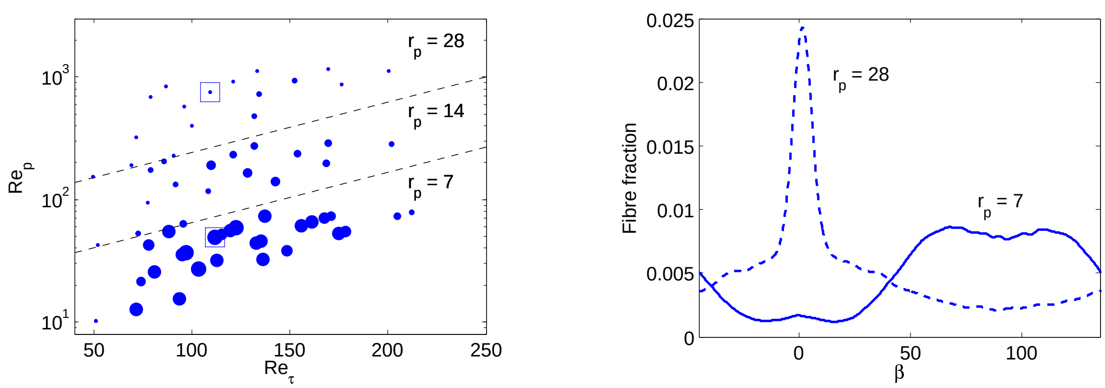
* Wallenberg Wood Science Center, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

† Linné FLOW Centre, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

karlh@mech.kth.se, kvick@mech.kth.se, fredrik@mech.kth.se

Keywords: Fibre suspension flows, Fibre streaks, Wall bounded turbulent flow, Fibre orientation

The shear flow in the headbox of a paper former plays an important roll for the structure and properties of the final paper. The headbox accelerates the pulp by transforming it from a pipe flow to a wide sheet that is jetted out onto the dewatering wires, where the paper is formed. In order to investigate phenomena present in a headbox, a turbulent fibre suspension flow near a wall is investigated in this study. The experimental setup consists of a slightly inclined open rectangular channel made of glass with reservoirs upstream and downstream. A pump is used to transfer the suspension from the downstream to the upstream reservoir. The suspension flows down the inclined channel driven by gravity. LDV measurements show that the flow is turbulent and fully developed. Cellulose acetate fibres with a density $\rho_f = 1300$ kg/m^3 and an aspect ratio of $r_p = 7, 14$ and 28 are used. Due to sedimentation most of the fibres are located close to the wall. The friction Reynolds number Re_τ is varied between 50 and 230 and the rotational particle Reynolds number Re_p between 10 and 1000. Re_p is based on the wall shear stress and the fibre length. By analyzing images taken from beneath, through the glass bottom of the channel, fibres are detected using a steerable filter. The position and orientation of the fibres in the wall parallel plane are obtained. The fibres form streamwise oriented streaks for some values of Re_τ and Re_p. Streakiness and streak widths are obtained through correlation analysis. The streakiness is shown for different Re_τ and Re_p in figure 1(a). Bigger dots represent more apparent streaky structures. The widths of the fibre-streaks are compared with the empirical value of $\sim 50l^+$ for low velocity streaks in turbulent boundary layers, where l^+ is the viscous length scale. The results show that the fibre-streaks scale in the same manner as the viscous sublayer streaks in a turbulent wall bounded flow. The orientation distributions of the fibres is highly dependent on the aspect ratio of the fibres. Furthermore, fibres with aspect ratio $r_p = 28$ are mostly aligned in the flow direction and fibres with $r_p = 7$ are mostly aligned perpendicular to the flow direction, see figure 1(b). The fibres with aspect ratio $r_p = 14$ are oriented more homogeneously, i.e. this might be an intermediate state.



(a) (b)

Figure 1: (a) Streakiness, bigger dots represent a lower value of the minimum of the spanwise correlation and thus a higher degree of streakiness. The orientation distributions for the measurements indicated by the squares are shown in (b). (b) Orientation distributions for $r_p = 28$ at $Re_p \approx 750$, $Re_\tau \approx 110$ (dashed line) and $r_p = 7$ at $Re_p \approx 50$, $Re_\tau \approx 110$ (solid line). $\beta = 0$ is the streamwise direction.

Two-way coupled simulations of ellipsoidal particles suspended in a turbulent channel flow

Lihao Zhao & Helge I. Andersson
Department of Energy and Process Engineering
Norwegian University of Science and Technology

This paper aims to present a mixed Eulerian-Lagrangian approach to simulations of dilute suspensions of solid ellipsoidal particles in turbulent fluid flows. The focus is mainly on the turbulence modulations induced by the presence of ellipsoidal particles. To achieve this goal, based on the Newton's third law, a two-way force-coupling scheme is implemented in a Navier-Stokes solver in order to study the detailed force interactions between the ellipsoidal particles and the carrier fluid. The turbulence field is fully resolved by means of Direct Numerical Simulations (DNS) at a frictional Reynolds number 360 (based on the wall distance h). The translational and rotational motions of the particles are handled by a Lagrangian point-particle approach. The size of the particles is assumed to be smaller than the smallest eddies and the particle Reynolds number is below unity.

A case with translational response time 30 and particle aspect ratio 10 was selected and simulated with 5 million particles. The results with force-coupling indicate an expected increase of mean bulk velocity as compared with the unladen flow. Furthermore, the streamwise turbulent intensity is augmented whereas the spanwise turbulent intensity and, in particular, the Reynolds shear stress are attenuated compared with the unladen flow. By looking at the instantaneous velocity contours, we observed that the overall turbulence is suppressed, i.e. the small-scale eddies are damped, and the flow comprises larger eddies than a flow without particles.

FINAL PROGRAMME

DAY 2 – Thursday, April 7, 2011

SESSION 5		FLUID-PARTICLE INTERACTIONS
CHAIR		C. Marchioli
TIME	SPEAKER	TITLE
09:00 – 09:20	**F. Lundell**	**Triaxial ellipsoids in creeping shear at high rotational Stokes numbers**
09:20 – 09:40	**H. Niskanen**	**On the development of fibre orientation in jet-to-wire impingement**
09:40 – 10:00	**S. Dearing**	**Optical characterization of fibers suspensions in turbulent pipe flow**
10:00 – 10:20	**J. Olson**	**Experimental characterization of turbulent fibre suspensions with wall suction**
10:20 – 10:40	**A. Kartushinsky**	**Deposition of solid particles at streamlined surface in turbulent flow**

10:40 – 11:10 COFFEE BREAK

SESSION 6		POLYMERS AND DRAG REDUCTION
CHAIR		M. Reeks
TIME	SPEAKER	TITLE
11:10 – 11:30	**G. Bellani**	**Turbulence modulation effects by finite-size ellipsoidal particles**
11:30 – 11:50	**A. Moosaie**	**Numerical investigation of drag reduction in turbulent channel flow by rigid fibers using a direct Monte-Carlo method**
11:50 – 12:10	**R. Delfos**	**Experiments on drag reduction**
12:10 – 12:30	**E. de Angelis**	**Wall turbulence with rod-like polymers**
12:30 – 12:50	**J. Gillissen**	**Polymer elasticity and turbulent drag reduction**

12:50 – 14:00 LUNCH

SESSION 7		AGGLOMERATES AND PARTICLES WITH INTERNAL DYNAMICS
CHAIR		R. Powell
TIME	SPEAKER	TITLE
14:00 – 14:20	**M.W. Reeks**	**A simple shell model for the break-up of agglomerates in turbulent flows**
14:20 – 14:40	**M. Dietzel**	**Lattice-Boltzmann simulations for characterizing the behaviour of agglomerates with different morphologies**
14:40 – 15:00	**F. Toschi**	**Small and large droplets in turbulent flows**
15:00 – 15:20	**D. Hanstorp**	**Dynamics of micro-rods in micro-fluidic channels**

15:20 – 15:50 COFFEE BREAK

SESSION 8		FOCUS ON INDUSTRIAL APPLICATIONS
CHAIR		A. Soldati
TIME	SPEAKER	TITLE
15:50 – 16:10	**M. Fantoni (FANTONI)**	**Fiber problems in the fiberboard making process**
16:10 – 16:30	**T. Parssinen (METSO)**	**Fiber suspension flow and paper quality: the importance of research work to Metso**
16:30 – 16:50	**J. Hamalainen**	**Presentation of COST Action on “Fiber suspension Flow modelling”**
16:50 – 17:30		**COST Action Preliminary Meeting**

Triaxial ellipsoids in creeping shear at high rotational Stokes numbers

F. Lundell*†

* Wallenberg Wood Science Center, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

† Linné FLOW Centre, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

fredrik@mech.kth.se

Keywords: triaxial ellipsoid, creeping shear, particle inertia

The fundamental problem of motion of triaxial particles in creeping shear is studied theoretically and numerically. The aim is to provide a basis for understanding of particle motions observed in various shear flow. In particular, the effect of particle inertia is studied. The analytical torques of Jeffery (1922) are coupled to the equations of rotation for a triaxial ellipsoid; the same methodology was used to study the motion of spheroids by Lundell and Carlsson (2010). Particle inertia induces a drift towards rotation around the shortest axis. The instabilitiy of this motion for inertia-free near-spheroids found by Hinch and Leal (1979) are here found to be stabilized for high enough particle inertia (measured by the Stokes number $St = \rho_p \dot{\gamma} l^2/\mu$ where ρ_p is the density of the particle, $\dot{\gamma}$ is the rate of shear, l is the particle length and μ is the dynamic viscosity of the suspending fluid). Thus, the chaotic motion found for some inertia-free triaxial ellipsoids by Yarin et al. (1997) are stabilized for strong particle inertia (compare figure 1 (c) and (d,e))

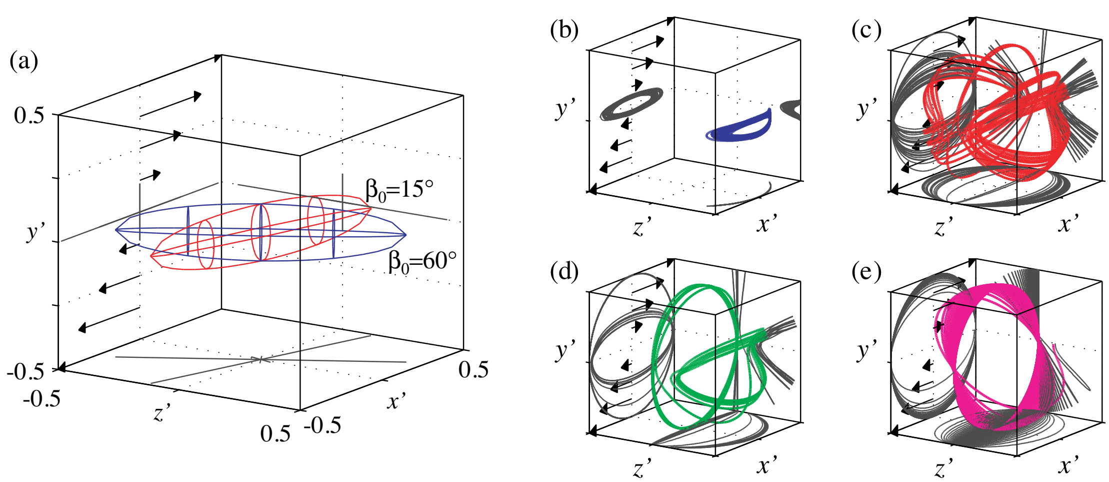


Figure 1: (a) Ellipsoids (illustrating different initial conditions) in a shear flow. (b,c) Colourcoded orbits of the particle endpoint for the two initial orientations shown in (a) for $St = 10$. (c,d,e) Orbits of the particle endpoint for $St = 10$, 100 and 1000. In (c), the particle shows a chaotic motion whereas (d,e) show a drift to a periodic motion.

References

E.J. Hinch and L.G. Leal. Rotation of small non-axisymmetric particles in a simple shear flow. *J. Fluid Mech.*, 92:591–607, 1979.

G. B. Jeffery. The motion of ellipsoidal particles immersed in a viscous fluid. *Proc. Roy. Soc. London*, 102(715):161–179, 1922.

F. Lundell and A. Carlsson. Heavy ellipsoids in creeping shear flow: Transitions of the particle rotation rate and orbit shape. *Phys. Rev. E*, 81: 016323, 2010.

AL Yarin, O Gottlieb, and IV Roisman. Chaotic rotation of triaxial ellipsoids in simple shear flow. *J. Fluid Mech.*, 340:83–100, Jan 1997.

On the development of fibre orientation in jet-to-wire impingement

H. Niskanen* and J. Hämäläinen**

*University of Eastern Finland, Department of Physics and Mathematics
P.O. Box 1627, FI-70211 Kuopio, Finland
e-mail: heidi.niskanen@uef.fi

**Centre of Computational Engineering and Integrated Design (CEID), Lappeenranta University of Technology
P.O. Box 20, FI-53851 Lappeenranta, Finland

ABSTRACT

Papermaking is a complex combination of several unit processes. One of the key processes is taking place in the so-called wet-end of the paper machine or more precisely, in a component called the headbox which is used to generate a thin, even jet. Headbox and its design affects notably on the paper properties. Its effect is huge especially on the fibre orientation, which contributes e.g. to the strength properties of the paper. The fibre orientation is largely controlled by the fluid dynamics, i.e. the flow acceleration and turbulence, of the headbox and its slice channel. From the headbox slice channel the process continues as the jet hits the moving fabrics i.e. the wire-section where the water removal is initiated. The fact that the jet enters into the open air and hits the wires cause great changes in the velocity profile and thus, plays important role in the development of the orientation profile. This naturally affects further the orientation of the fibres in the paper sheet. In this work, our focus is in the modelling of the fibre orientation in the jet-to-wire impingement.

As mentioned, the motion of the fibres is determined by the flow profile of the headbox and consists basically of translation and rotation. The orientation of the fibres is defined by the flow configuration in a way that the mean flow aligns the fibres into the flow direction whereas the velocity fluctuations distract the alignment of the fibres. In this work, we use a rather common approach (see e.g. [1-3]) to model the development of the fibre orientation. This approach involves a diffusion-convection equation which with proper definition of boundary conditions and flow properties is used in solving the orientation distribution of the fibres in a given flow conditions. Besides the velocity of the flow, the model accounts for turbulence effects with the diffusion coefficients. The model has been previously validated with experiments performed in a contracting channel geometry [4]. Here we use that model in order to predict the development of the fibre orientation in the jet and further in the jet-to-wire impingement. The results provide an insight how the jet and the jet-to-wire impingement modifies the orientation distribution.

References:

[1.] Olson, J., 2001. The motion of fibres in turbulent flow, stochastic simulation of isotropic homogenous turbulence. Int. J. Multiphase Flow 27, 20832103.

[2.] Hyensjö, M., 2008. Fibre orientation modelling applied to contracting flows related to papermaking. Ph.D. Thesis, Royal Institute of Technology, Stockholm.

[3.] Krochak, P., Olson, J., Martinez, D., 2010. Near-wall estimates of the concentration and orientation distribution of a semi-dilute rigid fibre suspension in poiseuille flow. J. Fluid Mech. 653, 431462.

[4.] Niskanen, H., et al. On the orientation probability distribution of flexible fibres in a contracting channel flow. Int. J. Multiphase Flow (2010), doi:10.1016/j.ijmultiphaseflow.2010.11.006

Euromech Colloquium 513, Dynamics of
non-spherical particles in fluid turbulence
6-8 April 2011, Udine, Italy.

Optical characterization of fibers suspensions in turbulent pipe flow

Stella Silvana Dearing and Alfredo Soldati *
Dipartimento di Energetica e Macchine, Università di Udine, Udine, 33100, Italy

Suspensions of elongated rigid fibers in turbulent flows are commonly encountered in applications of engineering interest, and may exhibit complicated rheological properties depending on the spatial distribution and orientation of the fibers. Despite the practical importance of fiber suspensions there is insufficient experimental data to validate numerical simulations and provide benchmarks. This paper presents an image analysis algorithm used to calculate orientation and distribution of fibers suspended in turbulent pipe flow. The algorithm is validated using artificial images. These images represent three-dimensional randomly orientated ellipsoids illuminated by a laser sheet and projected onto a two-dimensional plane. The error magnitude on the orientation distribution and number density is found by means of Monte Carlo simulations. Experiments are carried out considering small control volume near the pipewall. Results indicate that fibers exhibit preferred spatial orientation close to the pipewall and more randomized orientation close to the centerline, in qualitative agreement with the available numerical simulations.

*Corresponding Author: Email: soldati@uniud.it; Phone: +39 0432 558020; Fax: +39 0432 558027

H. Salem, S. Mokamati, S. Delfel, J.A. Olson & D.M. Martinez

We visualize experimentally both the concentration and velocity fields of a fibre suspension in the vicinity of wall with suction. The wall is considered rough with repeating contours machined onto the surface. Under dilute conditions we characterize the flow field with both particle image and laser Doppler velocimetry at various combinations of the ratio of the crossflow to suction velocities and compare these to CFD estimates. Both steady state and time dependent simulations were performed. In addition, a series of experiments were conducted to both map the concentration field near the vicinity of the slots using high speed video imaging and estimate the trajectory of individual tracer fibres.

Euromech Colloquium 513, Dynamics of
non-spherical particles in fluid turbulence
6-8 April 2011, Udine, Italy.

DEPOSITION OF SOLID PARTICLES AT STREAMLINED SURFACE IN TURBULENT FLOW

A. Kartushinsky, Y. Rudi, I. Shcheglov, S. Tisler, and M. Hussainov
Research Laboratory of Multiphase Media Physics, Faculty of Science,
Tallinn University of Technology, Tallinn, Estonia
aleksander.kartusinski@ttu.ee

Gas-solid particles two-phase flows met in a variety of engineering applications ranging from pneumatic conveying to fluidized bed reactors are accompanied by deposition of solid particles at various surfaces. The understanding of physical mechanisms that govern the deposition is essential for modeling of natural phenomena occurring and optimal design of industrial processes (Kartushinsky et al., 2009).

The influence of parameters of turbulent two-phase horizontal flow on the deposition velocity of solid particles were studied experimentally and theoretically for the horizontal flat plate.

Experiments were carried out in horizontal rectangular channel for the flow velocity of 5.1 m/s. Turbulence of gas flow was generated by the grid, and its intensity was about 3% along the test section. 12 and 23 μm corundum particles were applied in experiments. The quantity of particles deposited along the plate surface was determined by the high-speed imaging technique. The concentration and dynamic parameters of particles brought to the surface were determined by the PTV technique described in Hussainov et al. (2008).

The numerical simulation of the particles deposition was realized under both conditions - laminar and turbulent boundary layers developed near the flat plate surface. The flow conditions were characterized by magnitudes of the velocity of carrier fluid in free stream which was varied from 5 till 15 m/s for the given length of the plate 0.5 m. The behavior of gas and solid particles was approximated by the two-fluid model, or co-existed flow model, taking into account the viscous drag and gravity force factors, which were considered within the one-way coupling, similar to the model by Hussainov et al. (1995). The effect of the form of non-spherical particles was considered by change of factor of drag coefficient.

Analysis of the obtained results has shown that the deposition velocity strongly depends on the conditions of the two-phase boundary layer generated near the surface and the material and size of particles.

References

Kartushinsky, A.I., Krupensky, I.A., Tisler, S.V., Hussainov, M.T., Shcheglov, I.N. Deposition of solid particles in laminar boundary layer on a flat plate. High Temperature, Vol. 47, pp. 892–901 (2009).

Hussainov, M., Kartushinsky, A., Rudi, Y., Tisler, S., Shcheglov, I., Dispersion of Solid Particles in Grid-Generated Turbulent Flow. ANSYS Workshop "Multiphase Flows - Simulation, Experiment and Application", Dresden, Germany, 24 – 26 June 2008, CDROM (2008).

Hussainov, M., Kartushinsky, A., Mulgi, A., Rudi, Y., Tisler, S., Experimental and theoretical study of the distribution of mass concentration of solid particles in the two-phase laminar boundary layer on a flat plate, Int. J. Multiphase Flow, Vol. 21, 6, pp. 1141–1161 (1995).

Turbulence modulation effects by finite-size ellipsoidal particles

G. Bellani*, E. A. Variano†

* KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

† Civil and Environmental Engineering, University of California Berkeley, Berkeley (CA), USA

bellani@mech.kth.se, fredrik@mech.kth.se

Keywords: Isotropic turbulence, PIV

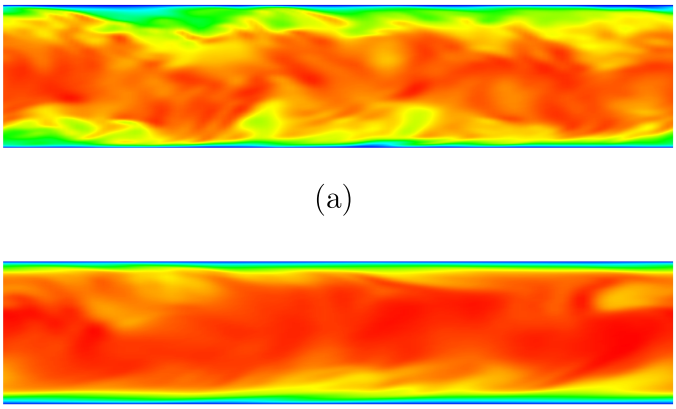
In this work we study turbulence modulation effects by rigid particles in homogeneous isotropic turbulence experimentally. We perform Stereoscopic Particle Image Velocimetry (SPIV) on both the fluid phase and the portion of the index-of-refraction matched particle on the plane illuminated by the laser sheet. These measurements yields the fluid-phase velocity and, thanks to a novel particle tracking algorithm, linear and angular velocities of the particles. Here we present statistics about the fluid phase. In particular we focus the discussion on the modification of the turbulent kinetic energy and of the turbulent spectra by particles of size in the range of the Taylor scales. Experiments have been performed with both spherical and ellipsoidal particles and spectral-pivoting has been observed for both cases.

Numerical investigation of drag reduction in turbulent channel flow by rigid fibers using a direct Monte-Carlo method

Amin Moosaie* and Michael Manhart
Fachgebiet Hydromechanik, Technische Universität München
Arcisstr. 21, 80333 München, Germany
Phone: +49-89-28922432, Fax: +49-89-28928332

Abstract

Addition of a small amount of fibrous or flexible polymers can dramatically reduce the frictional drag in turbulent wall flows. This effect is of technical interest in liquid transportation, e.g. oil pipe systems and fire fighting devices. Detailed numerical investigation of the drag reducing effect of polymeric additives has become feasible in the last decade, and has shed some light on the physical mechanisms of drag reduction. In this work, numerical study of the fiber-induced drag reduction in turbulent channel flow at $Re_\tau = 180$ is presented. The flow field is computed using direct numerical simulation (DNS) of Navier-Stokes equations in an Eulerian frame. The fibers are considered in a Lagrangian frame using a particle tracking scheme. The fiber conformation is computed by a direct Monte-Carlo solver. The Eulerian DNS and the Lagrangian Monte-Carlo solvers are two-way coupled. The Monte-Carlo method has the advantage that it does not require any closure model. The Lagrangian treatment of the suspended fibers is chosen because the underlying equation is purely convective, which is difficult to solve in an Eulerian frame without introducing numerical diffusion. We present the numerical algorithm and discuss the results, including first- and second-order statistics and the vorticity field.



(a)

(b)

Figure 1: Streamwise velocity contours of (a) Newtonian and (b) drag-reduced flows.

*Electronic address: `a.moosaie@bv.tum.de`; URL: `http://www.hy.bv.tum.de`

EXPERIMENTS ON DRAG REDUCTION BY FIBRES IN TURBULENT FLOW

R. Delfos, J. Hoving, J. Westerweel, B.J. Boersma
Process and Energy Technology Department
Delft University of Technology, Leeghwaterstraat 17, 2628 CA Delft, the Netherlands

INTRODUCTION

In turbulent pipe flow of liquids, drag reduction by polymer additives has been studied already for a few decades both experimentally and theoretically/numerically. Theoretical attempts to explain the physical working mechanism point towards the formation of an 'elastic layer' between viscous sublayer and logarithmic layer. Maximum drag reduction (MDR, 'Virk's asymptote') is found when it grows towards the full pipe cross section.

For gases, applications to reduce frictional drag are sought in the addition of small rigid fibres. A complication in the modelling of these fibres is that they have macroscopic dimensions compared to the smallest length scales of turbulence.

Experimental studies with fibres as additives are scarce, and the data found in literature is in general more scattered. In this paper, it is our aim to do a systematic study on friction drag of turbulent flow with monodisperse high-aspect ratio fibres.

EXPERIMENT & RESULTS

To minimise complications caused by particle inertia we performed experiments on frictional drag with water as the carrier fluid in a minimal volume horizontal flow-loop. We used nylon fibres of two diameters, i.e. $d = 10\&20\mu$m and four different fibre lengths, i.e. l, ranging from 0.5 till 4 mm, with this varying fibre aspect ratio $r = l/d$, between 25 and 400. Furthermore, we varied the bulk Reynolds number, $Re = U_B.D/\nu_{carrier}$ between $\approx 5.10^3$ and $\approx 2.10^5$, and the fibre volumetric concentration c, from 0 until values with increased drag. We start with a low fibre concentration c and measure pressure drop, hence drag difference relative to data with water only, over a relevant range of Reynolds numbers. Then we increase the fibre concentration c and so on.

With increasing concentration, we observe three regions. For a specific fibre, (here the 10 μm, $\times$ 1 mm), we find, as shown in figure 1:

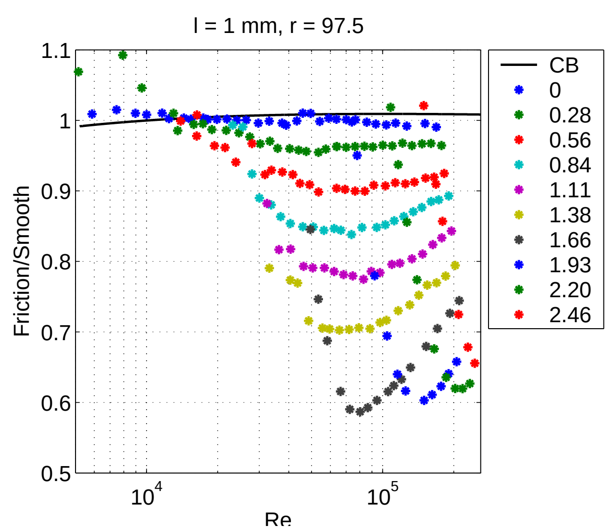
- At low concentrations, there is drag reduction DR at the middle range of Re, with maximum reduction a certain Reynolds number Re_{max}. For all Re, DR is roughly linear with fibre concentration c.
- At a certain concentration c_{MDR}, maximum drag reduction MDR is found at a certain Reynolds number $Re_{MDR} \approx Re_{max}$.
- Further increasing c, the change of drag with Re stays similar; however, the DR vs. Re curves now move towards higher Re, without a significant change in magnitude or shape. Further increasing c drag increases and eventually the flow loop gets clogged.

Compiling the data of experiments with seven different fibre lengths and diameters, we obtain the following:

- Re_{max} decreases significantly with increasing fibre length l, but does not depend on fibre diameter d. Best correlation is found with viscous wall units, as at Re_{max}, $l = 75\nu/u^*$.
- MDR decreases with increasing fibre length l. The 0.5 mm gives an MDR of more than 50%, the 4 mm fibres only 25%. The required concentrations c_{MDR} varies strongly: For the 4 mm fibre as little as 0.15% is needed; for the thick 0.5 mm fibre as much as 5%. The transition correlates well with the theoretically inspired $c_{MDR} \times r^2/(\ln(r) - 0.8) = 38 \pm 4$.
- Flow visualation shows that the flow changes dramatically around c_{MDR}. For low c, the flow is turbulent throughout the tube cross section; whereas beyond $c_M DR$ the fibres suppress internal motion: We get a plug flow near the core of the tube. This matches the observation bu Xu (2003), who obtained a flat velocity profile near the pipe centre.

Our experiments have given quantitative data on drag reduction in turbulent pipe flow with fibre additives, and have marked some different flow regimes, with a remarkable transition towards plug flow. Further investigation with more detailed experimentla techniques is required.

Based on these experiments and the derived scaling rules, application of fibres for natural gas transport pipe drag reduction seems unrealistic.



WALL TURBULENCE WITH ROD-LIKE POLYMERS

E. De Angelis[1], E.S C. Ching[2]
1 DIEM, Università di Bologna, 47100 Forlì, Italy
2 Department of Physics and Institute of Theoretical Physics, Chinese University of Hong Kong. Shatin,Hong Kong

It is a relatively a long time since the first simulation of drag reduction by polymers have been reproduced numerically, where velocity field obtained has successfully reproduced the well known drag reducing characteristics of such flows. With respect to the flexible case the simulations for rod-like polymers started only recently due to the higher complexity of the modellization. In particular, the data we will discuss have been obtained by using an appropriate, but very simplified, model for the rod-like polymers. As customary, the total stress tensor is obtained superposing the polymer stress tensor, T^p_{ij}, to the Newtonian contribution. with dynamic viscosity μ, and a part due to Specifically, in this approach each rod is represented by a neutrally buoyant axisymmetric particle, whose configuration is given in terms of the vector n_i. Once introduced the moments $\mathcal{R}_{ij} = \langle n_i n_j \rangle$, $\mathcal{R}_{ijkl} = \langle n_i n_j n_k n_l \rangle$ the field equation for the covariance matrix reads

$$\frac{\partial \mathcal{R}_{ij}}{\partial t} + u_k \frac{\partial \mathcal{R}_{ij}}{\partial x_k} = K_{ir}\mathcal{R}_{rj} + \mathcal{R}_{ir}K_{jr} - 2E_{kl}\mathcal{R}_{ijkl} - 6\gamma_B \left(\mathcal{R}_{ij} - \frac{\delta_{ij}}{3} \right) \tag{1}$$

where K_{ij} and E_{ij} are the velocity gradient and its simmetric part, respectively, and γ_B is the Brownian rotational diffusion of the rods. For the stress tensor instead, again following[1] we have

$$T^p_{ij} = \mu_p \left[E_{kl}\mathcal{R}_{ijkl} + 6\gamma_B \left(\mathcal{R}_{ij} - \frac{\delta_{ij}}{3} \right) \right] \tag{2}$$

μ_p is the rodlike polymers contribution to viscosity depending on the number density, which is proportional to the zero shear viscosity. In the present work we will consider a case where the Brownian diffusion term can be neglected, i.e. $\gamma_B \to 0$. Let's note that in this limit the polymer model depends only on the parameter μ_p. In the simulations the simple closure hypotesis will be used, i.e. $\mathcal{R}_{ijkl} = \mathcal{R}_{ij}\mathcal{R}_{kl}$.

Results obtained by simulations on a channel flow at different Reynolds number and concentrations will be discussed.

References

[1] Doi M., Edwards S.F. *The Theory of Polymer Dynamics*, (Oxford, New York) 1988.

Euromech Colloquium 513 Dynamics of non-spherical particles in fluid turbulence, Udine, Italy, 6-8 April 2011

Polymer Elasticity and Turbulent Drag Reduction

J. J. J. Gillissen
Department of Mult-Scale Physics
J.M. Burgers Centre for Fluid Mechanics
Delft University of Technology
Prins Bernhardlaan 6
2628 BW Delft
The Netherlands
j.j.j.gillissen@tudelft.nl

It is well-known that polymer drag reduction is positively correlated to extensional viscosity, which is coupled to polymer extension. Rod-like polymers always assume this favorable conformation, while randomly-coiling chains need to be unraveled by fluid strain rate in order to become effective. The coiling and stretching of flexible polymers in turbulent flow produces an additional elastic component in the polymer stress. The effect of the elastic stresses on drag reduction is unclear. To study this issue we compare Direct Numerical Simulations of turbulent drag reduction in channel flow using rod-like polymers and randomly coiling chains.

When compared at constant cr^2 both simulations predict the same amount of drag reduction. Here c is the polymer volume fraction and r is the polymer aspect ratio, which for flexible polymers is based on average polymer extension at the channel wall. This demonstrates that polymer elasticity plays a marginal role in the mechanism for drag reduction.

513th Euromech Colloquium on "Dynamics of non-spherical particles in fluid turbulence"
Udine,Italy, April 6th – April 8th, 2011

A simple shell model for the break-up of agglomerates in turbulent flows

Yasmine Ammar [1,2] and Michael W. Reeks[1]
[1]School of Mechanical and Systems Engineering, Newcastle University, UK
[2] Thermal-Hydraulics Laboratory, Nuclear Energy and Safety Department,
Paul Scherrer Institute, CH-5232, Villigen PSI, Switzerland

We describe and evaluate a stochastic model for the break-up of an agglomerate in a turbulent flow in which the agglomerate is subjected to a random sequence of straining and rotating flows simulating the Kolmogorov small dissipating scales of turbulent motion that are responsible for the agglomerate breakup. The agglomerate is considered to be composed of shells of primary particles, the total breakup rate of the agglomerate depending on the breakup rate of each shell forming the agglomerate. The breakup is thus very similar to the onion peeling mechanism for break up by erosion. In this context, of particular relevance is the modelling / simulation of multi layer resuspension of particles attached to surfaces in a turbulent boundary layer in which the multilayer resuspension is related to the resuspension rate from a single layer [1] . The latter is based on a primary removal rate constant $p(\xi)$ for the escape of particles from a surface adhesive potential denoted by a parameter ξ, the total removal rate being an integration over a broad range of adhesive potential i.e. values of ξ within the layer due to a distribution of particle sizes, micro-scale roughness and flow exposure. The removal rate of particles from any given layer depends upon the rate at which particles are exposed to the flow from the layers above. In the simplest model it is directly related to the rate of removal from the layer above. So that if $n_i(\xi,t)d\xi$ is the number of particles in layer i with adhesive and flow exposed between parameters ξ and $\xi + d\xi$ with probability density $\psi(\xi)$

$$\begin{aligned}\frac{\partial n_i(\xi,t)}{\partial t} &= -p(\xi)n_i(\xi,t) + \psi(\xi)\int_0^\infty p(\xi')n_{i-1}(\xi',t)d\xi' \\ &= -p(\xi)n_i(\xi,t) + \psi(\xi)\Lambda_{i-1}(t) \qquad (1)\end{aligned}$$

where $\Lambda_{i-1}(t)$ is the removal rate of particles from the layer above.

The model we propose for agglomerate breakup is a simple shell model in which the agglomerate is composed of a number of concentric spherical shells, each shell containing a number of primary particles which along with the local volume fraction are related to the fractal dimension of the agglomerate. In determining the way particles are bonded to together within each layer, we make use of the coordination number, i.e., the number of particles a given particle is in contact with. This varies according to the shape and distribution of primary particle size within the agglomerate. In our first approximation, the coordination number is two, which means that the particles are bonded to a particle in the layer above and below it.

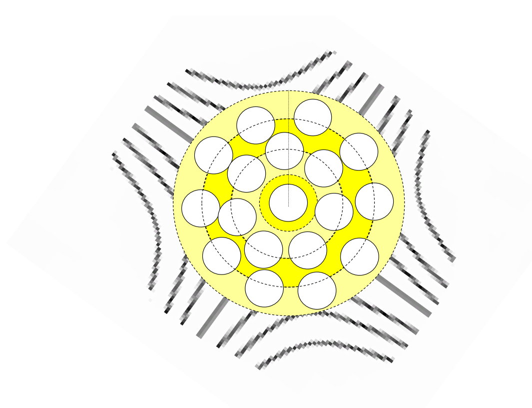


Figure 1: Shell model for agglomerate breakup showing the agglomerate at the centre of a symmetric straining flow

The determination of the single shell primary break-up rate constant, $p(\xi)$ is based on previous work on modelling the resuspension of particles attached to surfaces in a turbulent boundary layer [2].

The agglomerate sits within a random symmetric straining flow as shown in Figure 1, the strain rate having a prescribed Gaussian distribution with the the principal axes of the strain state radomly rotating to produce a statistically isotropic flow field. As in resuspension models for mono layer coverage, the mechanism of removal is based on the rocking and rolling of the particles about their asperity points of contact and as such the the moments derived from the drag force are assumed to be the principal breakup mechanism within each layer. The adhesive force is selected from a log normal distribution, being reduced by a factor ∼100 due to the roughness from that for smooth contact.

We first contrast the predictions for breakup based on 3 different models for the primary removal rate constant $p(\xi)$ and use them to make some predictions for the breakup of agglomerates in turbulent flow in a pipe comparing them with experimental measurements for the outer radius of the suspended agglomerates as a function of flow rate. Figure1 shows some results for the break-up of an agglomerate for which the coverage coefficients are assumed to be unity and for which the source of particles in any layer depends only on the removal of particles from the layer above.

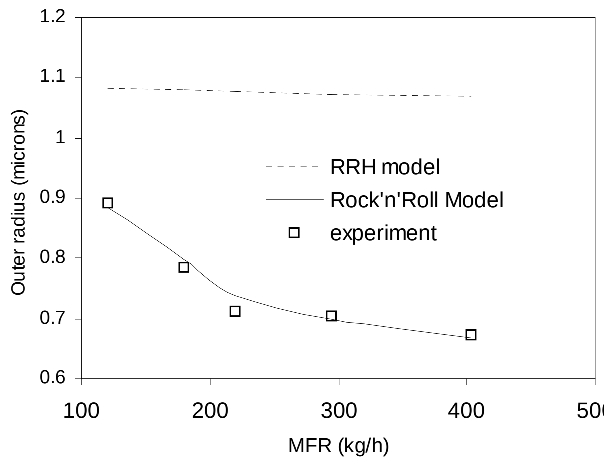


Figure 2: Breakup of agglomerates in turbulent pipe flow : experimental results versus model predictions

References

[1] Friess, H. and G. Yadigaroglu, Modelling of the resuspension of particle clusters from multilayer aerosol deposits with variable porosity. Journal of Aerosol Science, 2002. 33(6), 883-906.

[2] Reeks, M.W. and D. Hall, Kinetic models for particle resuspension in turbulent flows: Theory and measurement. Journal of Aerosol Science, 2001. 32(1), 1-31

Lattice Boltzmann simulations for characterizing the behaviour of agglomerates with different morphologies

M. Dietzel[1] and M. Sommerfeld[2]

Zentrum für Ingenieurwissenschaften, Martin-Luther-Universität Halle-Wittenberg,
D-06099 Halle (Saale), Germany
E-mail [1] : mathias.dietzel@iw.uni-halle.de
E-mail [2] : martin.sommerfeld@iw.uni-halle.de

Keywords: Lattice Boltzmann method (LBM), agglomerates, porous particles, drag coefficient, fractal dimension

ABSTRACT

Porous particles are present in various technical applications throughout many branches of process industry as well as in the field of medical treatment. Particle properties such as the aerodynamic diameter, the particle shape and the morphology highly influence their transport behaviour. Besides experiments, numerical simulations are capable of providing information about the physics that influence the dynamical behaviour of porous particles in fluid flow. Understanding and predicting the corresponding multiphase flows requires the study of the relationship between particle structure and particle motion. By performing fully-resolved direct numerical simulations (DNS) of single complex particles correlations between particle properties and their fluid dynamic behaviour are to be established to improve existing numerical models such as the Euler/Lagrange approach.

In this work, the lattice Boltzmann method (LBM) is used to simulate the flow around porous spherical agglomerates. Based on a random-driven distribution of the primary particles different agglomerate structures are realized. By modifying number, size or size distribution of the primary particles the porosity of the agglomerates is adjusted. Additionally, the mass distribution inside the agglomerate can be influenced to provide either a constant or a varying particle porosity along the agglomerate diameter. The characterization of the particle morphology is done using the radius of gyration, the fractal dimension and the porosity based on a convex hull around the particle.

From the flow simulation with LBM the fluid dynamic forces acting on the agglomerates can be determined (Dietzel and Sommerfeld 2008, Hölzer and Sommerfeld 2009). Thereto, the agglomerates are introduced into a laminar flow at different low Reynolds numbers (Figure 1). The particle boundaries are resolved using local grid refinement (Crouse 2003) realizing a high discretization near the particle surface and a coarser grid far from the particle. Based on the calculated forces as well as on equivalent diameters drag, lift and momentum coefficients are derived and compared for different particle

morphologies. Correlations incorporating structural characteristics of the agglomerates on the one hand and the fluid dynamic coefficients on the other hand are to be established.

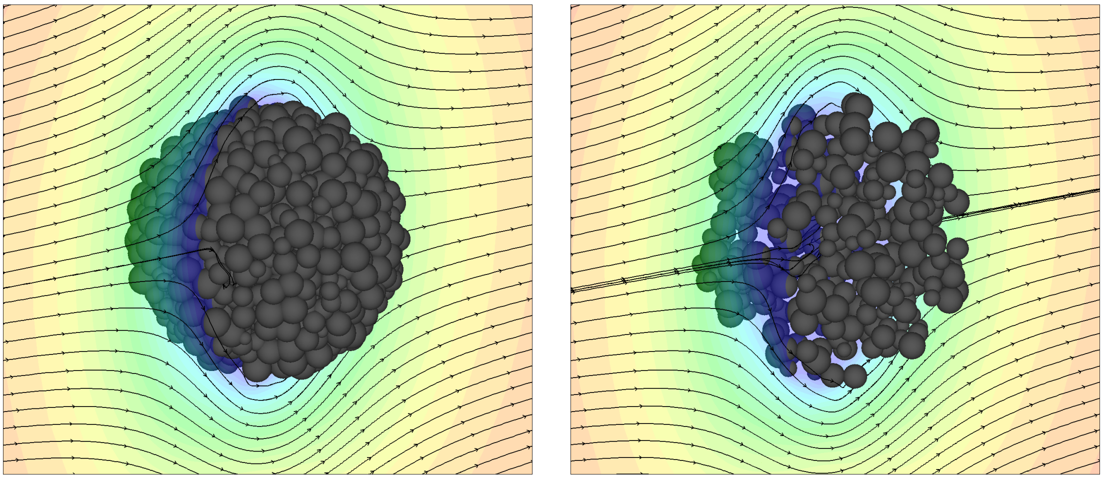


Figure 1 Comparison of fluid velocities in main flow direction at a cross-sectional plane through spherical agglomerates at Reynolds of 0.3:
left: 1400 primary particles, 30 % porosity; right: 350 primary particles, 80 % porosity

REFERENCES

Crouse, B., 2003: Lattice-Boltzmann Strömungssimulationen auf Baumdatenstrukturen; Dissertation, Technische Universität München.

Dietzel, M., Sommerfeld, M, 2008: Determination of aerodynamic coefficients of agglomerates using the Lattice-Boltzmann-Method; Proceedings of the 6th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries, Trondheim, Norway.

Hölzer, A., Sommerfeld, M., 2009: Lattice Boltzmann simulations to determine drag, lift and torque acting on non-spherical particles; Computers & Fluids, vol. 38, no3, pp. 572-589.

Euromech Colloquium 513, Dynamics of
non-spherical particles in fluid turbulence
6-8 April 2011, Udine, Italy.

Small and large droplets in turbulent flows

Federico Toschi[1], Luca Biferale[2], Prasad Perlekar[1], Mauro Sbragaglia[2]

[1]Eindhoven University of Technology, The Netherlands

[2]University of Tor Vergata, Roma, Italy

Turbulent emulsions are of relevance to many Natural and industrial flows alike. In order to study the statistical properties of droplets deformation and breakup in turbulence we perform high resolution numerical simulations of a multicomponent flow composed by two fluid with equal density. We aim at investigating the interplay between turbulent fluctuations and surface tension. The flow is solved in a cubic periodic box with a stirring at the largest scales in order to realize an homogeneous and isotropic turbulent flow field. The numerical simulations are performed by means of a fully-parallel Lattice Boltzmann code where the two fluid components are described by means of a Shan-Chen model without need for explicit interface tracking. Our numerical experiment allow to investigate e.g. the probability distribution function of droplet radii and the physics of the exchange of energy between surface and fluid fluctuations. We present preliminary results for a selected number of problem parameters.

Dynamics of micro-rods in micro-fluidic channels

J. Einarsson, A. Oladiran, P. Anderson, D. Hanstorp, and B. Mehlig

Abstract

We study the dynamics of micro-rods advected in the laminar flow field of a micro-fluidic channel (cross-sectional area 400 x 200 micron^2).
The micro-rods were produced by emulsification of a polymer solution under shear. Their lengths are of the order of 100 micrometer with aspect ratios typically 20. By means of a microscope we follow the orientational dynamics of individual rods.
We analyse their tumbling motion.

DAY 3 – Friday, April 8, 2011

SESSION 9		ADVANCES IN MEASUREMENT AND SIMULATION TECHNIQUES
CHAIR		R. van Hout
TIME	SPEAKER	TITLE
09:00 – 09:20	**M. Barri**	**New scheme for torque coupling**
09:20 – 09:40	**B. van Wachem**	**Modelling of gas-solid turbulent flows with non-spherical particles**
09:40 – 10:00	**G. Kondora**	**Drag coefficient of finite cylinder at moderate Reynolds number and its implementation to fibre model**
10:00 – 10:20	**G.M. Cartland Glover**	**Numerical models used for the modelling of the transport of fibrous insulation debris**
10:20 – 10:40	**A. Abbasi-Hoseini**	**Combined PIV and fibre orientation measurements on the KTH water-table**

10:40 – 11:10	COFFEE BREAK

SESSION 10		RHEOLOGICAL MODELLING
CHAIR		F. Toschi
TIME	SPEAKER	TITLE
11:10 – 11:30	**B. Mehlig**	**Poincare indices of rheoscopic visualisations**
11:30 – 11:50	**M.G. Rasteiro**	**Rheology of fibre suspensions: using the rheological characterization in CFD models for fibre flow**
11:50 – 12:10	**S. Haavisto**	**Rhelogical properties of a microfibrillated cellulose suspension**
12:10 – 12:30	**H.L. Pécseli**	**Numerical studies of encounter rates and transit times for small moving surfaces in turbulent flows**
12:30 – 12:50	**F. Paparella**	**Heavy spheroidal particles as a model for Phytoplancton cells**

12:50 – 13:00	Closing

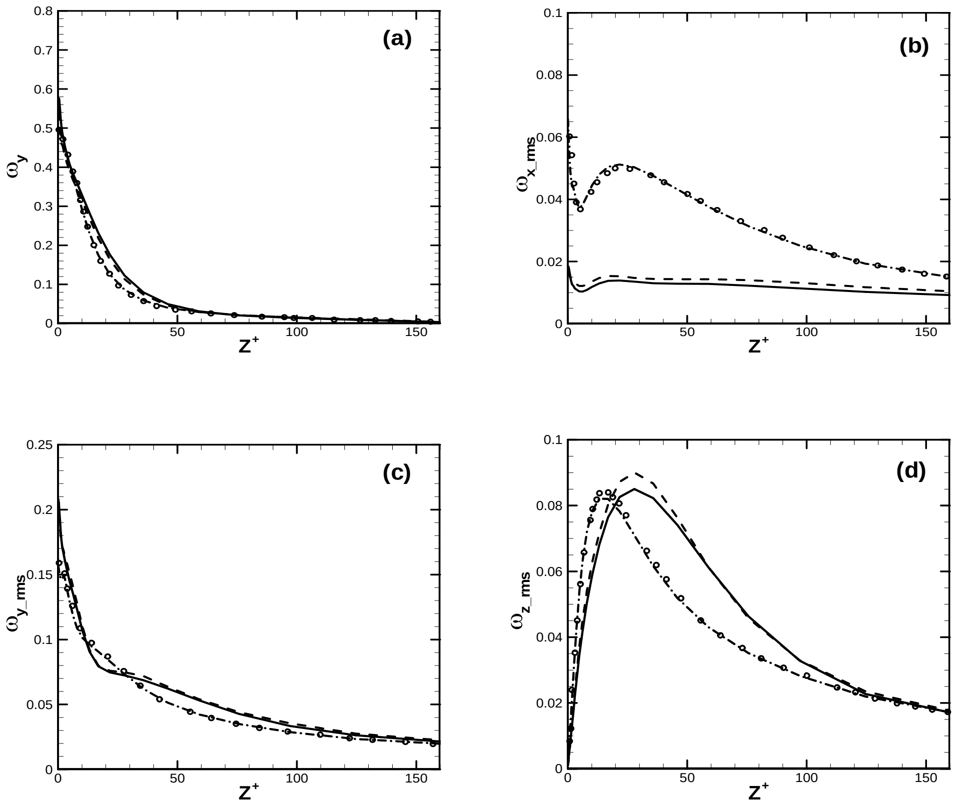
12:50 – 14:00	LUNCH

A new scheme for torque coupling

Helge I. Andersson, Lihao Zhao & Mustafa Barri

Department of Energy and Process Engineering,
The Norwegian University of Science and Technology, NO-7491, Norway

A novel scheme for the complete coupling between Lagrangian point-particles and a continuous fluid phase has been developed. A full mechanical coupling can only be achieved if torque-coupling is employed along with the more conventional force-coupling. The torque vector acting from the particles on the fluid is expressed in terms of a new *particle stress tensor* which adds to the Newtonian stress tensor. A fully-coupled simulation of spherical particles in a turbulent channel flow demonstrated that the resulting particle spin became strikingly different from that observed in the one-way coupled simulation by Mortensen et al. (2007). Sample results for ellipsoidal particles will also be presented at the Colloquium.



Mean spanwise particle spins and rms values of particle spins in streamwise, spanwise and wall-normal direction. One-way coupling by Mortensen *et al.* (2007) (open circle); force- and torque-coupling (solid line); force-coupling (dashed line); torque-coupling (dash-dot line).

Mortensen, P.H., Andersson, H.I., Gillissen, J.J.J. & Boersma, B.J. 2007 Particle spin in a turbulent shear flow. *Phys. Fluids* **19**, 078109.

Modeling of gas-solid turbulent flows with non-spherical particles

Berend van Wachem, Marian Zastawny, George Mallouppas, Fan Zhao

Department of Mechanical Engineering, Imperial College London, Exhibition Road, London SW7 2AZ, UK

B.van-Wachem@imperial.ac.uk

Keywords: Non-spherical particles, DNS, LES, Immersed boundary method, Particle-laden channel flow

In this contribution, we have developed a computational model to simulate the behaviour of turbulent flows laden with non-spherical particles. The computational model comprises different novel elements. Firstly, the drag, torque and lift relations of each particle shape is determined by means of DNS, where the particle is represented by the mirroring immersed boundary method (1). A large number of simulations is performed for each particle, to gather data on the drag, torque, and lift on the particle under various Re numbers and angles of attack.

Secondly, the resulting relations are then used in a fully coupled point-particle approach of a horizontal channel flow using the Large Eddy Simulation (LES) framework. The horizontal channel flow properties are based on the experimental and modeling work of (2). The Re number of the flow through the channel is around $Re \approx 22,500$, the particle equivalent diameter is $d_p = 200\,\mu m$ and the mass loading of the particles is $m = 1.0$. Three types of particles were studied: two types of ellipsoids and disk-shaped particles.

Thirdly, because of the high mass loading, a novel collision model to deal with the collisions between non-spherical particles and the particles and the wall is constructed based upon a Quaternion approach (3).

Simulations were performed of these 3 shapes of particles and compared to simulations with spherical particles and the available experimental data. Figure 1 shows some preliminary results. The results show there is a big effect of particle shape and particle-orientation. The effect of the walls on the particle orientation is shown in the figure below on the right. Also, the concentration profiles differ significantly, due to the variation in upward effect of the turbulence. Finally, the effect of wall roughness one the particle flow and orietation is researched.

References

[1] A. Mark and B. van Wachem, “Derivation and validation of a novel implicit second-order accurate immersed boundary method,” *Journal of Computational Physics*, vol. 227, pp. 6660–6680, 2008.

[2] J. Kussin and M. Sommerfeld, “Experimental studies on particle behaviour and turbulence modification in horizontal channel flow with different wall roughness,” *Experiments in Fluid*, vol. 33, pp. 143–159, 2002.

[3] S. Johnson, J. Williams, and B. Cook, “Quaternion-bases rigid body rotation and integration algorithms for use in particle methods,” *Int. J. Num. Meth. Engng*, vol. 74, pp. 1303–1313, 2008.

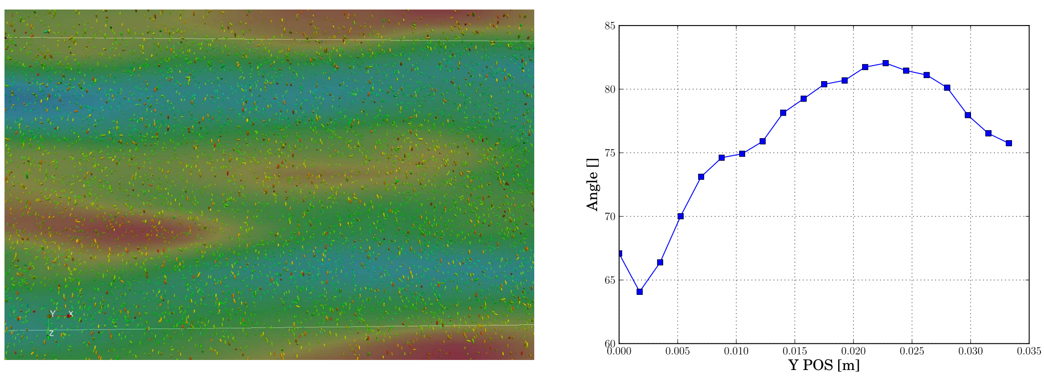


Figure 1: Left: Snapshot of the flow and the ellipsoids in the channel. Right: The angle of orientation of an ellipsoid in the flow; the wall has a large effect due to collisions.

Drag coefficient of finite cylinder at moderate Reynolds number and its implementation to fibre model

Grzegorz Kondora, Dariusz Asendrych
kondora@imc.pcz.czest.pl, darek@imc.pcz.czest.pl

Institute of Thermal Machinery, Czestochowa University of Technology
al. Armii Krajowej 21, 42-201 Czestochowa, Poland

Keywords: Fibre suspension, papermaking, drag coefficient, particle-level simulation

Abstract

In papermaking the modelling of fibre suspensions is of great importance. When the properties of final product are to be predicted the knowledge about fibres behaviour, in particular fibres orientation, is required. With Lagrangian approach, the most adequate simulation technique, single fibre is modelled as a chain of segments, which may be represented by either cylinders or prolate spheroids or, in the simplest case, spheres. The choice of element shape is determined by the desired fibre model properties as well as computational performance. When spheres are used to discretize the fibre, drag force distribution may be easily implemented from literature. However, with this segment shape one may expect huge computational cost (proportional to the number of segments) as well as the problems with fibres overlapping and collision detection. These problems may be overcome when fibre is constructed from cylinders. However, for isolated cylindrical particle drag force coefficient becomes a function of orientation angles (Fig. 1a) and these relationships are unknown (expect for creeping flow conditions, i.e. Re<<1).

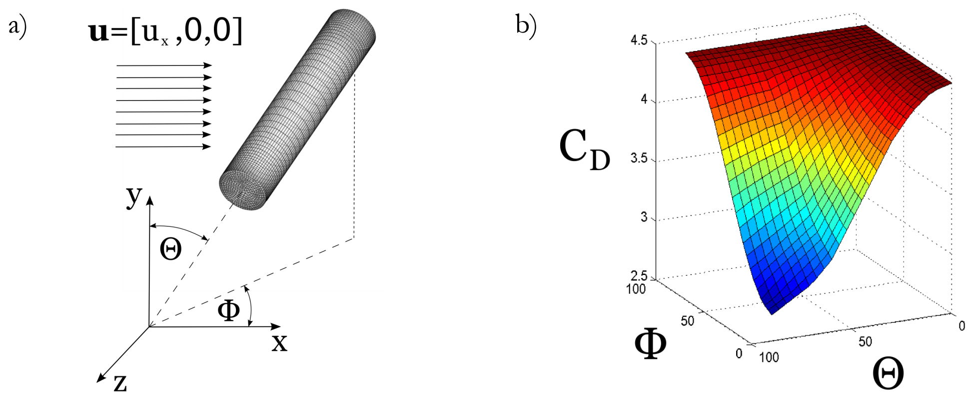


Fig.1. Orientation angles definition (a) and drag force coefficient vs orientation angles (b)

In present paper numerical algorithm has been developed enabling for detailed studies of drag force coefficient as a function of orientation angles, cylinder aspect ratio and Reynolds number. Automatic procedure allowed to generate numerical grid (Gambit) and then to conduct CFD calculations (Ansys Fluent). A sample distribution of drag force coefficient c_D for Re=5 is shown in Fig. 1b. The results were found to be fully consistent with available reference data [1].

Drag force and torque coefficients distributions were implemented to the previously developed Lagrangian fibre model and tested in simple shear flow. Results of simulations were compared with existing literature data including both computational as well as experimental ones.

Reference

1. Vakil, A. and Green, S. I.: Drag and lift coefficients of inclined finite circular cylinders at moderate Reynolds numbers. Computers & Fluids 2009; 38:1771–1781

Numerical models used for the modelling of the transport of fibrous insulation debris

G. M. Cartland Glover[1], E. Krepper[1], S. Renger[2], A. Seeliger[2], W. Kästner[2] and H. Kryk[1]
[1] Helmholtz-Zentrum Dresden-Rossendorf, Institute of Safety Research, P.O. Box 510119, D-01314, Dresden, Germany
[2] Hochschule Zittau/Görlitz, Institute of Process Technology, Process Automation and Measuring Technology, P.O. Box 1455, D-02754, Zittau, Germany
g.glover@fzd.de, e.krepper@fzd.de, srenger@hs-zigr.de and ASeeliger@hs-zigr.de.

Abstract

Mineral wool insulation material applied to the primary cooling circuit of a nuclear reactor maybe damaged in the course of a loss of coolant accident (LOCA). The insulation material released by the leak may compromise the operation of the emergency core cooling system (ECCS), as it maybe transported together with the coolant in the form of mineral wool fiber agglomerates (MWFA) suspensions to the containment sump strainers, which are mounted at the inlet of the ECCS to keep any debris away from the emergency cooling pumps. In the further course of the LOCA, the MWFA may block or penetrate the strainers. In addition to the impact of MWFA on the pressure drop across the strainers, corrosion products formed over time may also accumulate in the fiber cakes on the strainers, which can lead to a significant increase in the strainer pressure drop and result in cavitation in the ECCS. Therefore, it is essential to understand the transport characteristics of the insulation materials in order to determine the long-term operability of nuclear reactors, which undergo LOCA.

An experimental and theoretical study performed by the Helmholtz-Zentrum Dresden-Rossendorf and the Hochschule Zittau/Görlitz[1] is investigating the phenomena that maybe observed in the containment vessel during a primary circuit coolant leak. The study entails the generation of fiber agglomerates, the determination of their transport properties in single and multi-effect experiments and the long-term effects that particles formed due to corrosion of metallic containment internals by the coolant medium have on the strainer pressure drop.

The focus of this presentation is on the numerical models that are used to predict the transport of MWFA by CFD simulations. A number of pseudo-continuous dispersed phases of spherical wetted agglomerates can represent the MWFA. The size, density, the relative viscosity of the fluid-fiber agglomerate mixture and the turbulent dispersion all affect how the fiber agglomerates are transported. In the cases described here, the size is kept constant while the density is modified. This definition affects both the terminal velocity and volume fraction of the dispersed phases. Only one of the single effect experimental scenarios is described here that are used in validation of the numerical models. The scenario examines the suspension and horizontal transport of the fiber agglomerates in a racetrack type channel. The corresponding experiments will be described in an accompanying presentation (see abstract of Seeliger et al.).

Keywords: Mineral Wool Fiber Agglomerates, Loss of Coolant Accidents, Containment Sump, Computational Fluid Dynamics, Multiphase Flow

---

[1] The reported investigations are funded by the German Federal Ministry of Economics and Technology (BMWi) under Contract No. 1501360 and 1501363.

Combined PIV and fibre orientation measurements on the KTH water-table

A. Abbasi-Hoseini,[*] K. Håkansson,[†] M. Kvick,[†] F. Lundell[†‡] & H. I. Andersson[*]

[†] Department of Energy and Process Engineering, Norwegian University of Science and Technology, 7491 Trondheim, Norway

[†] Wallenberg Wood Science Center, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

[‡] Linné FLOW Centre, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

afshin.abbasi-hoseini@ntnu.no, karlh@mech.kth.se, kvick@mech.kth.se, fredrik@mech.kth.se, helge.i.andersson@ntnu.no

Keywords: PIV, fibre orientation, turbulent channel flow

Velocity and fiber orientation distributions on the KTH water table are studied. On the water table, a water (or suspension) film is flowing down an inclined plate (2000 $\times$ 560 mm) driven by gravity. This is a good experimental model of one half of channel flows often used in direct numerical simulation studies. The friction Reynolds number (based on the liquid layer thickness, which corresponds to half the channel width in a channel flow) is $Re_\tau = 170$. Measurements are made by capturing images of fibres (cellulose acetate fibres, $d = 70\ \mu$ m and $l = 1$ mm) and PIV tracer particles in planes parallell to the wall. The planes are defined by a illumination with a laser sheet and images were acquired both with and without fibres. The images will be analyzed in two ways: PIV will be used to determine the local and instantaenous flow velocities and a steerable filter will be used to determine the fiber orientation and position of the fibres in the images. The first purpose is to provide information on local structures of flow, fibre positions and fibre orientations, albeit not simultaneous. When this has been achieved, efforts will be made to extract flow data from the images with both tracers and fibres by eliminating the fibres from the images once their positions and orientations has been determined.

Poincare Indices of Rheoscopic Visualisations

M. Wilkinson, V. Bezuglyy, and B. Mehlig

Abstract

Suspensions of small anisotropic particles, termed 'rheoscopic fluids', are used for flow visualisation. By illuminating the fluid with light of three different colours, it is possible to determine Poincare indices for vector fields formed by the longest axis of the particles. Because this vector field is non-oriented, half-integer Poincare indices are possible, and are observed experimentally. An exact solution for the direction vector appears to preclude the existence of topological singularities. However, we show that upon averaging over the random initial orientations of particles, singularities with half-integer Poincare index appear. We describe their normal forms.

Rheology of fibre suspensions: using the rheological characterization in CFD models for fibre flow

M. G. Rasteiro, F. A. P. Garcia, P. Ferreira
Chemical Engineering Department
University of Coimbra, Coimbra, Portugal
Contact email: mgr@eq.uc.pt

The rheological characterization of a wide range of pulp suspensions varying in morphology (short and long fibers) and consistency, was performed in a newly built rheometer [1] and models for their rheological behaviour were obtained [2].

This information was then introduced in a CFD model for fibre suspension flow in pipes, based on the k-ε turbulence model, developed using the COMSOL 3.5 software. The validating parameter for the model results was the pressure drop in the pipe, which had also been previously measured in a pilot rig, for the different types of suspensions.

The CFD model reproduced well the experimental data, though requiring an adjustment of both the turbulence length and intensity scales. The values retrieved for these two parameters could be correlated with the fibre characteristics and with the differences in concentration of the suspension, indicating a turbulence damping due to the presence of the fibres.

Future work will be directed to establishing quantitative relationships between the turbulence parameters and the suspension characteristics (fibre type and consistency) in order to render the model predictive.

1. Blanco, A., C. Negro, H. Fuente, J. Tijero, “Rotor selection for a Searle-type device to study the rheology of paper pulp suspensions”, *Chem. Eng. Process.*, **46** (1), 37-44 (2007).

2. Carla A.F. Ventura, A. Blanco, C. Negro, F.A.P. Garcia, P. Ferreira, M.G. Rasteiro, "Modelling Pulp Fibre Suspension Rheology", *Tappi J*, 6, 7, 17-23 (2007).

Rheological properties of a Microfibrillated Cellulose Suspension

Sanna Haavisto*, Martina Lille**, Johanna Liukkonen* and Juha Salmela*
Technical Research Centre of Finland, P.O. Box 1603, 40101 Jyväskylä, Finland
Addresses: * P.O. Box 1603, 40101 Jyväskylä, Finland. ** P.O. Box 1000, 02044 VTT, Finland
Email: forename.surname@vtt.fi

In this work a novel laboratory-scale pipe rheometer is utilized in rheological characterization of concentrated MFC and cellulose suspension. The method is based on a combination of pulsed ultrasound velocity profiling (UVP) and pressure difference measurement (PD).

The rheological properties of the suspensions are described in terms of viscosity and pressure loss. The results are compared to rotational rheometer results. It is demonstrated that well controlled pipe flow environment together with UVP-PD technique is efficient in characterizing the rheological flow behavior of complex slurries.

Numerical studies of encounter rates and transit times for small moving surfaces in turbulent flows

H. L. Pécseli and J. Trulsen

University of Oslo, Norway

The motion of passively advected particles is studied by numerical simulations of homogeneous and isotropic turbulent flows. Simultaneous trajectories of many (up to 400.000) passively moving point-particles are followed in time. Two problems are addressed: encounter rates and transit time distributions. Encounter rates are obtained by following small virtual surfaces with characteristic radii in the inertial as well as in the viscous range of the turbulence. For a Lagrangian description, we select one of these reference particles as the "center" of a self-consistently moving closed absorbing surface. The surface determines a reference volume, which may be a sphere, for instance. We subsequently obtain estimates for the time variation of the statistical average of the particle flux into this volume. The variation of the flux with the scale size of the surface of interception, as well as the variation with basic flow parameters, is well described by a simple model, in particular for radii smaller than a characteristic large length scale, the outer scale, for the turbulence. For very small radii the particle flux is given by Brownian motion. The length scale separating the turbulent fluxes for scales in the viscous subrange and Brownian fluxes is expressed in terms of the Schmidt number. We estimate also the probability distribution of the transit times of particles through the prescribed volume. The transit time is defined as the difference between entrance and exit times of surrounding particles advected through this volume by the turbulent motions. Simple scaling laws are obtained for the probability density of the transit times in terms of the basic properties of the turbulent flow and the geometry. The analysis is extended to include a finite reaction time, so that particles spending times shorter than some prescribed time-interval will not interact with the reference particle, but leave the reference volume, with the possibility of interacting at some later time. The reference volumes need not be spherical, also other shapes are considered. The numerical results are compared to similar observations from laboratory experiments, and good agreement has been demonstrated. The results of our analysis are relevant for describing some features of chemical reactions, but also for understanding details in the feeding rate of micro-organisms in turbulent waters, for instance.

Heavy Spheroidal Particles as a Model for Phytoplancton Cells

Francesco Paparella
Dip. di Matematica "Ennio De Giorgi", Università del Salento

Antonello Provenzale
Istituto di Scienze dell'Atmosfera e del Clima, CNR, Torino

December 13, 2010

Micro and nano phytoplanctonic cells show an extreme variety of morphologies, which, in most cases, are rather different from the spherical shape. They also are generally heavier than the water in which they live, therefore any turbulence or large-scale motion in the fluid affects their ecology in a crucial way.

Several results are known about the settling of spherical particles in fluids subject to either turbulent or laminar motion (e.g. [2]). Here we investigate the dynamics of heavy of spheroidal particles (both oblate and prolate) with numerical solutions of the equations proposed by Maxey [1], valid for very small particle Reynolds number. In particular, we compute the distribution of suspension times of spheroidal particles in cellular and other simple flows. We also discuss the separation of nearby particles: while spheres separate exponentially, the distance between two spheroids, for small separations and short time intervals, grows linearly, at a rate dependent on the mutual orientation between the particles.

We conclude with some considerations on the ecological and evolutionary significance of our findings.

References

[1] M. R. Maxey, "On the advection of spherical and non-spherical particles in a non-uniform flow," Philosophical Transactions: Physical Sciences and Engineering 333, no. 1631 (1990): 289–307.

[2] 1. Claudia Pasquero, Antonello Provenzale, and Edward Spiegel, "Suspension and Fall of Heavy Particles in Random Two-Dimensional Flow," Physical Review Letters 91, no. 5 (8, 2003).