Report of the Pilot Center Germany North

Coordinator: Prof. Dr.-Ing. M. Sommerfeld

Otto-von-Guericke University Magdeburg
Multiphase Flow Systems, Halle (Saale), Germany
Martin.sommerfeld@ovgu.de

1 Organizational Structure

The ERCOFTAC Pilot Center Germany North is currently composed of 18 members in total, 13 from university institutes, 4 members from industry, and one member from research centers. In 2011 the Pilot Center has been merged with the PC Germany West, and the University of Luxembourg has joined the new merged PC Germany North.

The center is coordinated by Prof. Dr.-Ing. Martin Sommerfeld from the Otto-von-Guericke University Magdeburg who also represents the Pilot Center and two special interest groups (SIG) in the Scientific Programme Committee. At the moment there is no industrial member from the PC Germany North involved in the Knowledge Network Committee.

2 Members

The PC Germany north has presently 18 members.

2.1 Universities

Helmut-Schmidt-Universit at Hamburg

- Professur für Strömungsmechanik

Hochschule Niederrhein, Krefeld
Hochschule Ostwestfalen Lippe, Lemgo

- Labor für Strömungsmaschinen und Fluiddynamik

Leibniz Universit at Hannover

- Institut für Turbomaschinen und Fluid-Dynamik (TFD)

Otto von Guericke-Universit at Magdeburg (OvGU)

- Fluid Dynamics and Technical Flows
- Mechanical Process Engineering
- Multiphase Flow Systems

RWTH Aachen

- Aerodynamisches Institut
- Institut für Technische Verbrennung
- Lehrstuhl für W arme und Stoffübertagung

Technische Universit at Berlin
TU Bergakademie Freiberg

- Institute of Mechanics and Fluid Dynamics

Universit at Luxemburg

2.2 Research Centers

Institute of Aerodynamics and Flow Technology, Braunschweig and Göttingen, German Aerospace Center (DLR)

2.3 Industries

Airbus Industries
Rolls-Royce Deutschland GmbH
Volkswagen AG
Viessmann Werke GmbH & Co KG

3 Activities of the Center

The following activities of the PC German north during the reporting period may be highlighted:

3.1 Technology Day

Annually the PC Germany North jointly with the PC Germany South is organizing a technology day which so far was held in Stuttgart. This day being comprised of presentations and discussion sessions is focused on different current research areas such as turbulence, fluid-structure interaction, complex fluid flows, flows in turbomachinery and multiphase flows. The technology day aims at enhancing the exchange between the research groups of the PCs and especially between Academia and Industry. Although ERCOFTAC is an ideal platform, the cooperation between Industry and Academia is still quite weak. This year's technology day (2017) is focussing on multiphase flows including participation from several University institutes and industrial partners as for example Bosch and ANSYS. Over the years normally around 100 scientists from all over Germany attend the technology day.

3.2 Workshop on Two-Phase Flow Predictions

The series of workshops started at the University of Erlangen and then was held at the University of Halle during the last 20 years. The workshop was regularly organized every three years mainly by Prof Sommerfeld, now at the University of Magdeburg (OvGU) Germany. The international workshop aimed at bringing together scientists from all over the world presenting recent development in modelling and numerical calculation of dispersed multiphase flows. An essential part of the workshop was furthermore the selection and definition of test cases for validating numerical computations including novel modelling approaches. Mostly experimental test cases were selected, which over the years considered more and more complex situations. During the last two workshops the following test cases were selected, to mention only a few:

- liquid-solid fluidized bed (IMFT Toulouse)
- pneumatic conveying of fine powder in a horizontal pipe (University Halle)
- dense particle-laden free jets with different mass loading (University of Florida)
- Dispersion of rod-like particles in a free jet (University Udine, University of Roma)

The simulation results of these test cases were compared, presented and discussed during the workshops. Two of these test cases are included in the QNET-ERCOFTAC data base (i.e. evaporating sprays and particle-laden swirling flows). The workshop normally had between 70 and 80 participants and is therefore an effective forum for scientific discussion.

3.3 Links to special interest groups

SIG 1: Large-Eddy Simulation
SIG 12: dispersed Turbulent Two-Phase Flows
SIG 15: Turbulence Modelling
SIG 28: Reactive Flows
SIG 33: Transition Mechanism, Prediction and Control
SIG 34: Design Optimisation
SIG 37: Bio-Fluid Mechanics
SIG 41: Fluid-Structure Interactions
SIG 46: Oil, Gas and Petroleum

4 Research Activity

In this section the reports on the research activities of presently active PC members are presented. All other members have not submitted an actual report:

4.1 Helmut-Schmidt-University Hamburg, Department of Fluid Mechanics

Fluid-structure interaction (FSI) is one of the top research activities of the department, since it plays an important role in many technical applications. The objective is the coupled simulation of large lightweight structures such as thin membranes (outdoor tents, awnings...) exposed to turbulent flows. For this purpose, a complementary experimental/numerical strategy is pursued. On the one hand, challenging FSI benchmark cases under clearly defined boundary conditions are developed and set-up in either a water or a wind tunnel. The ambitious task is to measure not only the fluid flow by optical contactless measuring techniques (PIV, V3V, LDA), but likewise the deformation or displacement of the flexible structure by digital image correlation or laser distance sensors. These benchmarks help to understand the complex physical phenomena of coupled FSI problems, but are also particularly valuable to validate and improve FSI codes. Therefore, the test cases and measured data sets are made available for all interested groups on the ERCOFTAC QNET-CFD Knowledge Base Wiki under: `http://qnet-ercoftac.cfms.org.uk/` (see UFR_2-13, UFR_2-14, and UFR_3-33).

On the other hand, to study these complex FSI problems based on simulations, a multi-physics code framework based on large-eddy simulation and specific

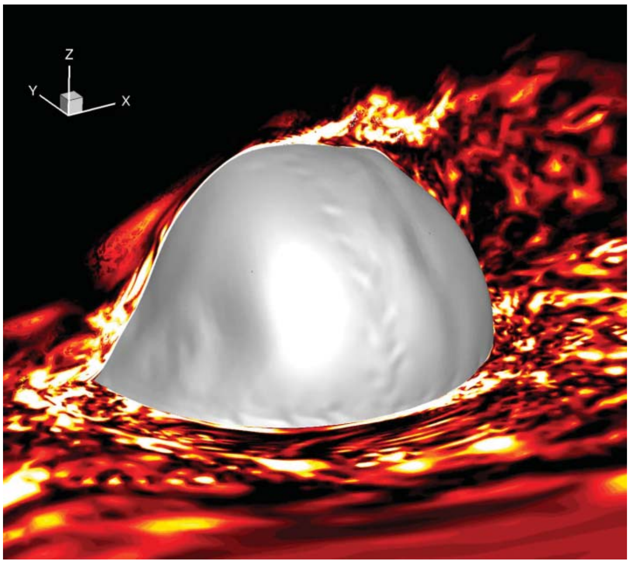

Figure 1: Turbulent flow around a flexible membranous structure (air-inflated hemisphere): Vorticity magnitude and displacements magnified by a factor of 15

shell/membrane elements was developed. It tackles the coupled problem applying a partitioned solution approach relying on a finite-volume solver for the fluid, a finite-element solver for the thin structures and a coupling interface (see application in the Figure). The stable and highly efficient FSI methodology running on high-performance computers such as SuperMUC is continuously improved for example by a new grid deformation strategy such as a recently developed hybrid IDW-TFI algorithm or by a synthetic turbulence inflow generator including wind gusts.

A second major research field is the numerical simulation of turbulent disperse multiphase flows, especially at high volume or mass loadings. For this purpose, an Euler-Lagrange code based on LES and a very efficient particle tracking scheme was developed. Originally, only particle-laden flows were tackled, but in the meantime also bubble-laden and droplet-laden flows can be predicted. For dense flows a four-way coupled approach is a must. In contrast to many previous studies relying on statistical collision models, in the present investigation a deterministic, but nevertheless efficient algorithm is applied for handling collisions. That builds the basis for recent developments of appropriate models for describing the agglomeration of dry, electro-statically neutral particles, the adhesion of such particles at bounding walls, the coalescence of microbubbles, and a composite collision outcome model for surface-tension driven droplet-droplet collisions. The models developed for these different physical phenomena and the entire simulation tool were carefully validated based on a variety of test cases and are now applicable to real applications.

A third ongoing research activity is related to the development of advanced hybrid LES-URANS methods. The main objective is to combine the advantages of LES and (U)RANS-based approaches in order to simulate wall-bounded turbulent flows with reasonable accuracy within acceptable simulation times. Recent improvements are concerned with an enhanced description of the near-wall flow based on exact relations for the dissipation rate. Furthermore, a source term based methodology was developed, which allows taking (artificially generated) inflow turbulence into account. The advantage of this technique is that it enables to inject inflow turbu-

lence in sufficiently resolved flow regions prohibiting that it is damped out before reaching the region of interest. The methodology was successfully tested based on the flow around a SD7003 airfoil with a laminar separation bubble, which disappears with increasing turbulence intensity in accordance with measurements.
More Information: `breuer@hsu-hh.de`

4.2 Hochschule Ostwestfalen Lippe, Lemgo, Fluiddynamics and Turbomachinery Laboratory

An improved design allows for the optimization of fluid dynamic processes, e.g. reversible turbomachines, jet impingement, confined two-phase flow. Based on the physical phenomena lab scale models for experimental investigations and simulation models for numerical investigations are derived. The results are correlated with dimensionless numbers. Based on these correlations an optimization is possible leading, to an increase in energy efficiency in industrial applications.

Most turbomachines have a distinct flow direction; nevertheless a reversible flow direction for axial impellers is desirable in certain circumstances in ventilation and drying applications or tidal power plants. In order for the impeller performance to be similar in both directions, the blade profiles have to conform to distinct symmetry conditions. As data for this kind of profiles is scarce, the flow around such reversible profiles is investigated. Most common are elliptical airfoils and S-shaped profiles. These are mostly derived from existing profiles using a twice cambered (i.e. S-shaped) camber-line. The flow over elliptical airfoils and S-shaped profiles is investigated numerically solving the two dimensional incompressible Reynolds averaged Navier-Stokes (RANS) equations. With an increase in camber and profile thickness lift and drag also increases. The most important parameter is the camber. The thickness distribution has a smaller influence. Due to the S-shape the flow is asymmetric and complex: There is always flow separation as well as reattachment and changes in the direction of the pressure gradient due to the curvature changes. For strong cambered profiles, these effects are predominant and the advantages using a twice cambered profile might diminish. Jet impingement and the resulting convective heat and mass transfer are the main mechanism in many cooling, heating and drying applications. A direct numerical simulation (DNS) or large eddy simulation (LES) may accurately describe the complex flow-phenomena, but they are limited to the simulation of single jets as they are computationally expensive. Configurations with multiple jets interacting and various operating conditions, as encountered in industrial applications, are described using RANS. The turbulence model used is important. The k-omega SST model is recommended but there are differences in the results with regard to the implementation of the model. For a configuration corresponding to the ERCOFTAC "Impinging jet" test case (QNET-CFD UFR 3-09) using the SST k-omega model without wall functions with the turbulence viscosity defined using the modulus of the mean strain rate tensor and a linear constitutive relation allows for a good agreement with the measured values.

As a model to describe the multiphase flow in a mixing chamber, the flow in a partially filled lid-driven cavity is investigated (see Figure). A two dimensional rectangular cavity is partially filled with liquid, one side wall is moving upward. Depending on the shape of the cavity, the fluid level and the Reynolds number different flow patterns evolve; a continuous liquid film on the wall (low Reynolds numbers) up to a foam and spray like two-phase flow (high Reynolds numbers). The flow is simulated using the Volume of Fluid (VOF)-method.

Figure 2: Flow in a partially filled cavity (water-black, air-white) at Re= 850. The wall on the right side is moving upwards for 0.85 s

4.3 Otto von Guericke-University Magdeburg, Fluid Dynamics & Technical Flows

The group's research activities cover a broad range of scientific problems and technical applications, relying on both advanced experimental measurements and on detailed simulations, and cannot be thoroughly described in such limited space. Interested readers are referred to our website (`http://www.lss.ovgu.de`) for more information. Most research projects focus on 1) multiphase reactors and processes, 2) optimization of turbomachines using our in-house optimization software OPAL++, 3) medical flows, or 4) turbulent reacting flows. Advanced experimental techniques (PIV, also high-speed, stereoscopic or tomographic; Particle Tracking Velocimetry; Laser-Induced Fluorescence; ...) and simulation tools (Direct Numerical Simulations; Lattice-Boltzmann approach; Immersed Boundaries; Discrete Element Models coupled with CFD; ...) are combined to support fundamental understanding and improve existing processes.

Concerning multiphase flows, our group considers mostly particulate flows, also in the dense regime (as found for instance in crystallizers or in classifiers), as well as bubbly flows (bubble columns and other gas-liquid reactors), and decanters. For turbomachines, energy generation from wind or water flows remains the focus of our investigations, together with the transport and separation of two-phase flows. Our research concerning medical flows concentrates on therapy improvement for damaged cerebral arteries, on turbulent flows in the respiratory system, and on efficient blood transport with reduced damage of red blood cells. Finally, research on reacting flows considers fundamental ignition processes, safety predictions, and instabilities in internal combustion engines burning a variety of fuels, as well as

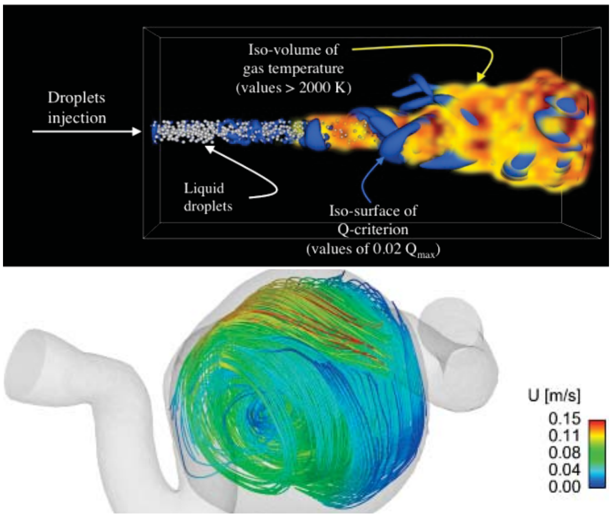

Figure 3: Direct Numerical Simulation of spray ignition in a turbulent flow (top); streamlines in a cerebral aneurysm phantom model obtained by Stereo-PIV measurements (bottom)

liquid-liquid mixing, gas-liquid mass transfer, and new reactor concepts.

For more information, contact `thevenin@ovgu.de`

4.4 Otto von Guericke-University Magdeburg, Mechanical Process Engineering

The research philosophy of Mechanical Process Engineering is to perform research across the scales, ranging from the research at the nanometer range, to the scale of machinery and apparatuses. At these various scales, we work on theory, develop numerical and modelling approaches, and perform experimental work. The focus of research is on multi-phase flows, particle technology, and computational engineering. Research topics range from understanding the behaviour and properties of an individual particle, the behaviour of thin film flows, to the large-scale modelling of gas-solid and gas-liquid flows. Research topics include: non-spherical particles, particles with rough surfaces, production and properties of nano-particles, physics of impacting particles, physics of liquid films and studying the dynamics of droplets.

The figure below shows an example of one of the research projects which has been carried out by our group: understanding the behaviour inside Wurster type fluidized bed coater. This project is concerned with understanding this process at the scale of the individual particle (e.g. shape, coating process), to developing large-scale models to predict the overall behaviour. The models have been developed by and validated with experimental work. The figure below shows a snapshot of the Wurste type fluidized bed coater. The colours on the particles indicate the thickness of the coating.

For more information `berend.vanwachem@ovgu.de`

Figure 4: Numerical simulation based on coupled CFD-DEM for a Wurster type fluidised bed coater

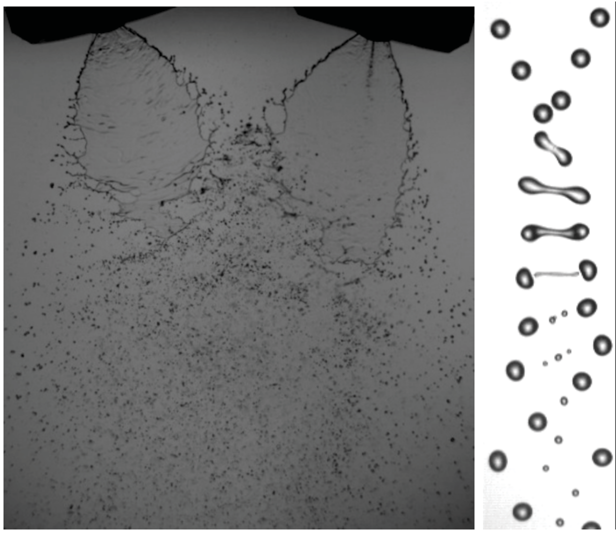

Figure 5: Fan spray interaction in a model spray dryer (left) for analysing high viscous droplet collisions as illustrated for stretching separation (right)

4.5 Otto von Guericke-University Magdeburg, Multiphase Flow Systems

The research at the group of Multiphase Flow Systems (MPS) of the Otto-von-Guericke-University (OvGU) is related to modelling of elementary processes in dispersed multiphase flows as a basis for numerical calculation of technical and industrial processes, mainly using the classical Euler/Lagrange approach. In this method, applicable to large scales, the flow field is calculated by RANS (Reynolds averaged Navier-Stokes equations) in connection with an appropriate turbulence model or by LES (Large eddy simulations) with sub-grid-scale models. The particulate phase (solid particles, droplets or bubbles) is modelled in a point-particle approximation allowing the consideration of a huge number of particles (several millions) in an industrial process. However, this approach requires that all processes occurring on the particle scale need to be described by appropriate models, as for example particle collisions, agglomeration, droplet coalescence and wall collisions including a possible deposition.

For developing these models experiments and direct numerical simulations (DNS) are conducted. The experiments are mainly based on high-resolution imaging

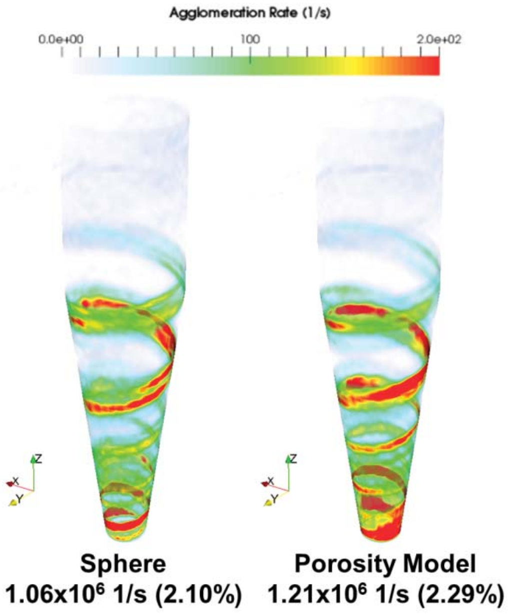

Figure 6: LES-Euler/Lagrange simulation of particle agglomeration in a cyclone. Due to the consideration of porosity in agglomeration modelling they grow larger and experience consequently more collisions with other primary particles. This yields improved separation of fine particles in a cyclone

techniques combining LED arrays or Lasers for illumination and high-speed cameras. Present studies are related to droplet collisions (Figure 5), solid particle-droplet collisions, non-spherical particle wall collisions, oscillating bubble motion near walls and dispersion of fibres in turbulent flows. DNS, mainly based on the Lattice Boltzmann method, is presently conducted related to the behaviour of oscillating bubbles, fibre dispersion and the agglomeration of fine particles as well as the motion of agglomerates in turbulent flows.

The resulting models are being implemented in the Euler/Lagrange method using the OpenFOAM platform. Present technical applications are related to pneumatic conveying, large-scale spray dryers (Figure 5), bubble columns, particle separation in gas cyclones including agglomeration (Figure 6), drug powder dispersion in inhaler devices and particle deposition in pulmonary airways. Figure 6 shows the calculated agglomeration rate in a cyclone comparing the standard volume equivalent sphere model to the advanced model considering porous and hence larger agglomerates. Through this size effect the agglomeration rate is higher for the porosity model.

For providing validation data also several technical-scale facilities are available, such as spray dryer, pneumatic conveying pipes system, mixing vessel and a water channel.

Further Information `martin.sommerfeld@ovgu.de`

4.6 RWTH Aachen University, Institute of Aerodynamics

The Institute of Aerodynamics performs experimental, theoretical, and numerical research in fluid mechanics. As to the numerical activities, a hierarchical unstructured Cartesian mesh method has been developed over the last couple of years. The approach enables the simulation of viscous flow interacting with freely moving boundaries. By using a cut-cell approach for the sharp resolution of the boundaries, the method exhibits a strict conservation of mass, momentum, and energy and a high accuracy. The solution-adaptive mesh refinement provides a flexible approach for high-resolution simulations of rigid-body motion in viscous flow. The discretization at the moving boundaries is applicable to complicated geometries and sharp edges appearing in technical applications. The numerical scheme for the coupling of the fluid and the structural motion was complemented by a new Runge-Kutta method which significantly reduces the costs for tracking moving boundaries and the subsequent reinitialization of the solver [1]. The accuracy and the efficiency of the new scheme were demonstrated for several three-dimensional flow problems.

In the following, two applications are briefly discussed. First, the results of the turbulent low Mach number flow through an axial fan at Reynolds number of 9.36×10^5 investigated by large-eddy simulation (LES) in [2, 3] are summarized. The focus in this numerical analysis was on the impact of the tip-gap size on the overall flow field. Special attention was paid to the vortical structures in the tip-gap region, whereas the small tip-gap possesses a permanent transition. A reduction of the tip-gap width reduces the amplitude of the tip-gap vortex wandering motion and increases the frequencies of the dominant modes, which was also confirmed by experimental data.

Furthermore, the modulation of decaying isotropic turbulence by 45,000 spherical particles of Kolmogorov-length-scale size was studied in [4]. It was shown that the particles illustrated in Fig.7 absorb energy from the large-scales of the carrier flow while the small-scale turbulent motion is determined by the inertial particle dynamics. Whereas the viscous dissipation rate of the bulk is attenuated, the particles locally increase the level of dissipation due to the intense strain rate generated near the particle surfaces due to the crossing-trajectory effect. Analogously, the rotational motion of the particles decouples from the local fluid vorticity and strain-rate field at increasing particle inertia. The high level of dissipation is partially compensated by the transfer of momentum to the fluid via forces acting at the particle surfaces. An analysis of the small-scale flow topology showed that the strength of vortex stretching in the bulk flow is mitigated due to the presence of the particles. This effect is associated with energy conversion at small wavenumbers and the reduced level of dissipation at intermediate wavenumbers.

Literature

1. Schneiders, L., Günther, C., Meinke, M., and Schröder, W., Journal of Comp. Physics, Vol. 311, pp. 62-86, doi: https://doi.org/10.1016/j.jcp.2016.01.026, 2016.

2. Pogorelov, A., Meinke, M., and Schröder, W., Phys. Fluids, Vol. 27, 075106, doi: 10.1063/1.4926515, 2015.

3. Pogorelov, A., Meinke, M., and Schröder, W., International Journal of Heat and Fluid Flow, Vol. 61(B), pp. 466-481, doi: https://doi.org/10.1016/j.ijheatfluidflow.2016.06.009, 2016.

4. Schneiders, L., Meinke, M., and Schröder, W., J. Fluid Mech., Vol. 819, pp. 188-227, doi: https://doi.org/10.1017/jfm.2017.171, 2017.

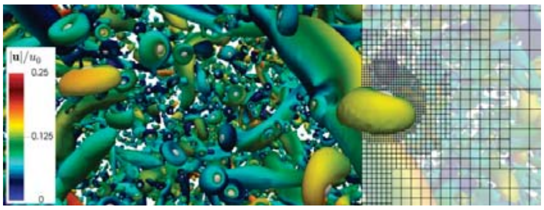

Figure 7: Direct particle-fluid simulation for 45,000 particles of Kolmogorov size; instantaneous vortical structures plus velocity distributions

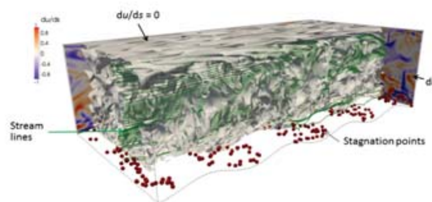

Figure 8: Direct particle-fluid simulation for 45,000 particles of Kolmogorov size; instantaneous vortical structures plus velocity distributions

4.7 RWTH Aachen University, Institute for Combustion Technology

Continuing our previous work, we have examined streamline segments in more detail. A streamline segment is defined as the arclength of a streamline bounded by a local minimum and local maximum of the magnitude u of the velocity, i.e. the isosurface $\partial u/\partial s = 0$ where $\partial/\partial s$ is the derivative along the streamline. Streamline segments can then be parameterized by the velocity difference Δu between the endpoints and the arclength distance l. Because streamlines depend on the underlying velocity field, examining statistics of streamline segments seems promising. Previous work of the institute had shown that the segment statistics are nearly universal for homogeneous isotropic turbulence. For instance, it has been found that the length distribution P(l) of streamline segments does not depend on the Reynolds number when normalized with the mean segment length lm. For that reason, we have looked at segment statistics in a plane channel flow and a channel flow with a wavy wall (see the figure below).

In case of the plane channel flow, three distinct regions can be identified, namely the turbulent core region, the logarithmic layer and the buffer layer. Both for the turbulent core region and the buffer layer, the same normalized length distribution P(l) as for homogeneous isotropic turbulence has been found, whereas the statistics differ in the buffer layer close to the wall. In case of the wavy wall channel flow, the statistics do not only depend on the wall-normal distance, but also on the slope of the wall, since the flow is also inhomogeneous in the downstream direction. Again, the length distribution of the streamline segments for the crest region, through region, expanding slope and contracting slope were found to collapse when normalized with the respective mean segment length lm. These observations imply that streamline segments are indeed suitable to analyse and compare statistics of turbulent flows in general, i.e. are not restricted to homogeneous isotropic turbulence.

4.8 RWTH Aachen University, Institute of Heat and Mass Transfer

WSA is proud to report on the extension of the collaborative research area 129 (SFB/TR 129) "oxyflame" by DFG. On July 1st 2017, oxyflame has started its second funding period, which is intended to run until June 2021. Within oxyflame scientists from the fields of gas and coal combustion, thermodynamics and chemistry work together in 19 projects. The collaborative research area is carried by 21 PI's coming from three universities (RWTH Aachen University, Ruhr University Bochum and Technical University Darmstadt). Oxyflame is aiming at the development of methods and models for oxyfuel combustion of solid fuels, here pulverized coal and biomass. For this mostly laser-based diagnostic methods are further developed and applied to the harsh conditions of pulverized fuel flames, while the accompanying modelling spans from atomistic ab initio modelling of in-particle reactions and highly resolved DNS to LES simulations of an entire combustion chamber.

While striving for a continuous scientific exchange with colleagues from all around the world, which often happens at an individual level, oxyflame has established a biennial international workshop as a platform for discussion with the entire scientific community. The 2nd international Workshop on Oxyfuel Combustion will be held in Bochum on February 14 and 15, 2018. Topics covered there are not strictly limited to oxyfuel combustion of solid fuels. Contributions which fit to the methods pursued and applied within oxyflame (www.oxyflame.de) also are highly welcome. Five key-note speeches (T. Faravelli, T. Fletcher, M. Alzueta, R. Axelbaum and V. Raj) form the backbone of the program, which covers oral presentations as well as a poster session. (Link to call for abstracts: `http://www.oxyflame.de/fileadmin/oxyflame/media/PDF-Dateien/Oxyflame_2ndWorkshop_Call.pdf`). For more information either use the links provided above or contact Reinhold Kneer (leading PI of oxyflame, kneer@wsa.rwth-aachen.de).

4.9 TU Bergakademie Freiberg, Institute of Mechanics and Fluid Dynamics

The research activities at the chair of Fluid Dynamics and Turbomachinery at TU Bergakademie Freiberg are focused on three main research areas: bubbly and particle-laden flows, fluid machinery, and granular fluid dynamics. Regarding bubbly and particle-laden flows, fundamental correlations of these flows are investigated, for example, the swarm behaviour or the influence of swirl on bubbly or particle-laden flows in experiments and numerical simulations. The results obtained are then used to analyse complex bubbly and particle-laden flows in various technological applications, for example bubble columns in metallurgy or classifier flow in the processing technology. Among others, ongoing projects are part of the collaborative research centers SFB 799 "Trip-Matrix Composite" and SFB 920 "Multi-Functional Filters for Metal Melt Filtration".

Detailed analysis of the flow and temperature fields in complete fluid machines, i.e. also in the mechanical components, form the core of the activities in this research area. Corresponding accurate data are required in order to make the best use of innovative materials or bionic principles, for example. The projects that are located here are usually carried out in close cooperation with various application partners. Funding for current projects is granted among others from industrial collective research IGF of the German Federation of Industrial Research Associations AIF.

The main aim of the research area granular fluid dynamics is the development of continuum mechanical models for the description of dense, granular flow processes, such as flows with dominant granular material and secondary fluid phases. The models are typically obtained by micro-macro-transition from simulation results of the discrete element method DEM on the micro (particle) scale. A recently granted project is part of the priority programme SPP 2005 "Opus Fluidum Futurum".

4.10 University of Luxembourg, Institute of Computational Engineering

The Institute of Computational Engineering (ICE) aims at building intuitive and interactive platforms for computational engineering problems that allow the users not only to understand and predict the behaviour of real systems but also to better capture the interaction between models and data and hence gain insights into unconventional and counter-intuitive phenomena. The institute is collaboratively led by Prof. Bernhard Peters, StÃľphane Bordas and Andreas Zilian. Within the institute Prof. Peters is responsible for the thermo-/fluids dynamics section and is head of the XDEM research team. The Extended Discrete Element Method (XDEM) is a novel and innovative numerical simulation technique that extends the dynamics of granular materials or particles as described through the classical discrete element method (DEM) by additional properties such as the thermodynamic state, stress/strain, or electromagnetic field for each particle. While DEM predicts the spatial-temporal position and orientation for each particle, XDEM additionally evaluates properties such as the internal temperature and/or species distribution. These predictive capabilities are further extended by an interaction to fluid flow by heat, mass and momentum transfer and impact of particles on structures. It requires a coupling to Computational Fluid Dynamics (CFD) and Finite Element Method (FEM) to cover a large range of engineering applications. Hence, the following strategic competences are established:

- Computational process engineering
- Computational material science
- Computational dynamics

While computational dynamics deals with various aspects of the discrete element method (DEM) such as non-spherical particles and high performance computing, computational process engineering and material science address a large number of aspects of thermal conversion of particulate material with a strong interaction to a fluid phase e.g. heat, mass and momentum exchange. Applications are as diverse as transport of debris, pharmaceutical industry, automotive, agriculture food and processing industry and engineering, construction and agricultural machinery, metals manufacturing, mining, biomedical, cement, energy production and additive manufacturing. The latter application is shown in the following figure 10 for fusion of a bed of metal powder. For predicting results, High Performance Computing (HPC) is of particular importance so that the entire simulation platform is parallelised. The latter is supported by a strong collaboration with Prof Bouvry of the department of computer science. Accessing two computer clusters promotes parallel computing based on the Message Passing Interface (MPI) and OpenMP, and thus, efficiently complements numerical and computational efforts to reduce CPU time. Prof. Peters is engaged in many European and international collaborations and represents the University of Luxembourg as a member in the European Research Community on Flow, Turbulence and Combustion (ERCOFTAC), International Association of Engineering Modelling, Analysis and Simulation (NAFEMS), European Federation of Chemical Engineers (EFCE), International Flame Research Foundation (IFRF), InterPore and is a honorary member of the TechnetAlliance.

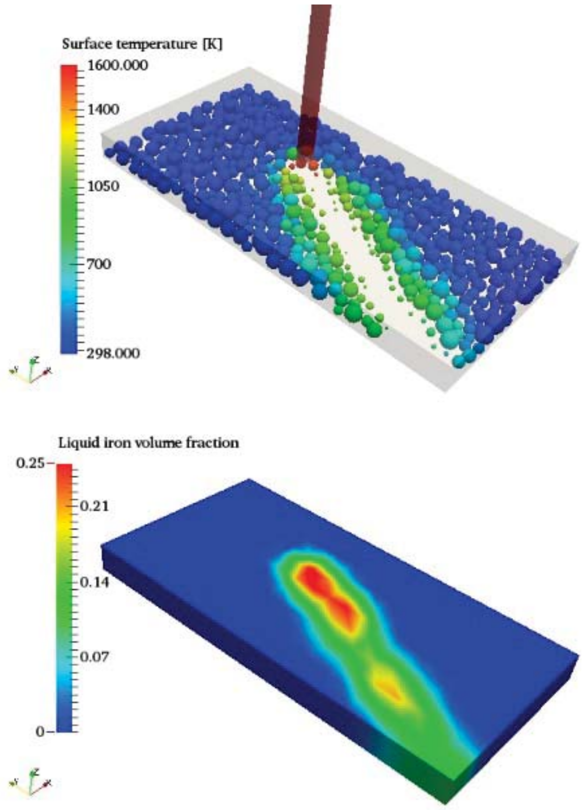

Figure 9: Laser beam traversing a packed bed of metal powder and heating up and melting individual particles of which the liquid mass fraction is transferred into a multiphase flow of gas and different liquids

4.11 Institute of Aerodynamics and Flow Technology, DLR

The Institute of Aerodynamics and Flow Technology is a leading research institute in the field of aerodynamics/aeroacoustics of aircraft and aerothermodynamics of space vehicles. It has two main sites in Braunschweig and Göttingen and a division in Cologne. In total more than 250 scientists are engaged in numerical and experimental research of air, space and ground vehicles with most of the physical modelling activities located in Göttingen.
The physical modelling activities aim at developing and improving turbulence and transition simulation capabilities, especially in the unstructured, finite-volume CFD codes TAU and THETA. The compressible DLR TAU code is used for the simulation of the external aerodynamics about air vehicles, while the incompressible DLR THETA code is applied for combustion problems and

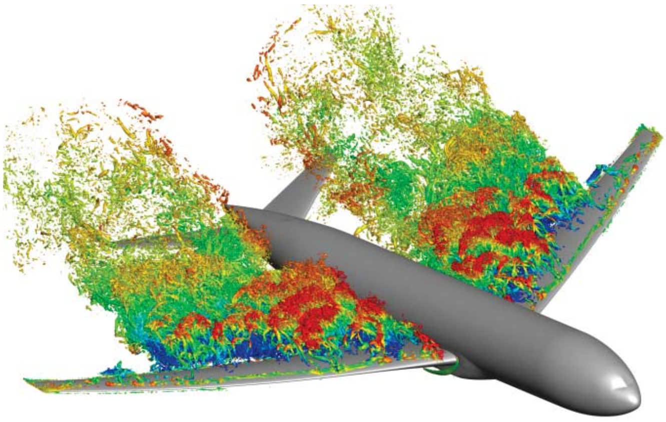

Figure 10: Generic aircraft configuration at subsonic stall simulated with the DLR TAU code using DDES as SRS approach; the Q-criterion reflects the small-scale turbulent structures coloured by the velocity

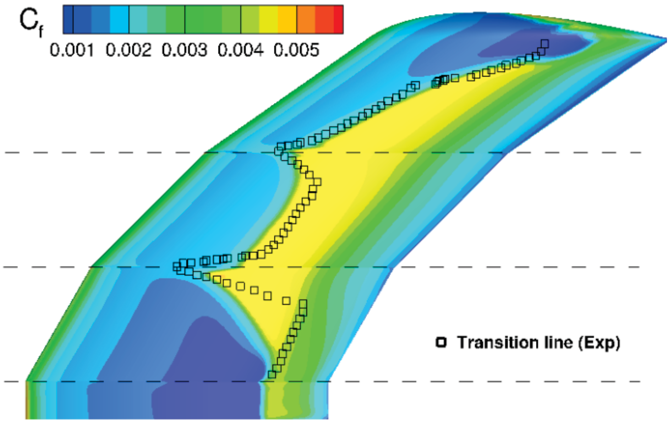

Figure 11: TU Braunschweig sickle wing at M = 0.16, $Re = 2.75 \times 10^6$ and $\alpha = -2.6°$ with experimental transition line and skin friction distribution simulated using the Re_{CF} transition model and the TAU-standard Reynold stress model

wind turbine applications. In the Institute of Aerodynamics and Flow Technology turbulence and transition modeling is organized along four development lines with focus on CFD development:

- Reynolds stress models (RSM): RSM are to be used as standard RANS approach for any kind of configuration, including highly complex industrial configurations.

- Scale-resolving simulations (SRS): SRS techniques, such as hybrid RANS-LES methods (HRLM), are provided for a targeted application for specific components of aircraft (Fig. 10) or military configurations.

- Transition prediction and modelling: The consideration of all relevant transition mechanisms and their impact within the simulation is a necessary condition for accurate results of turbulence models toward the borders of the flight regime (Fig. 11).

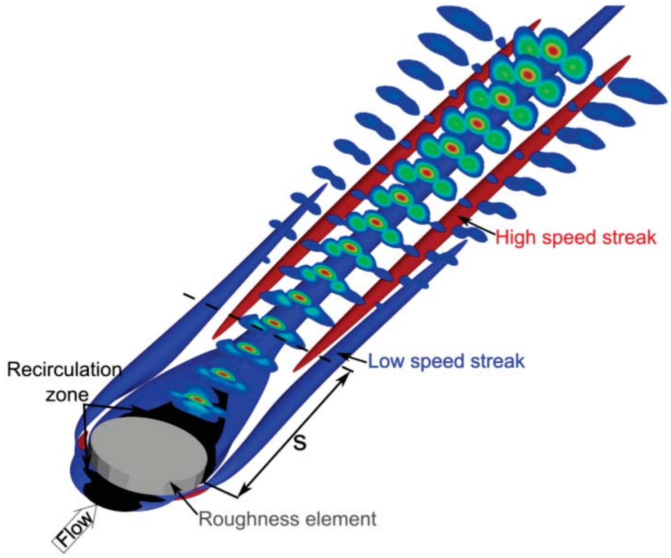

Figure 12: Visualization of the steady recirculation zones and the streaks in the laminar wake of a small cylindrical roughness together with a typical wake flow instability that can trigger premature laminar-turbulent transition in the wake flow

- Turbulence modelling improvements: Existing turbulence models are modified to improve prediction for certain flow phenomena, e.g. for adverse pressure gradient, by a physics-based extension of the calibration basis of model coefficients. For this, targeted experimental (physical & numerical) investigations for specific flow phenomena are carried out in order to find the corresponding scaling laws and to derive suitable model modifications.

A major future effort for the institute is the transfer of all existing modeling capabilities into the 'next-generation flow solver' of the institute, Flucs (FLexible Unstructured CFD Software), which is currently under development. Specific research activities in the field of transition prediction and modeling focus on an advanced understanding and improved modelling of the physical mechanisms of transition, the development of more reliable transition prediction tools and studies on advanced concepts for transition control. An emphasis is currently put on the detrimental effects of small surface irregularities like steps, gaps and 3-D roughness on the transition location aiming at a more reliable specification of allowable tolerances, e.g. for the surface quality of future laminar swept wings of transonic aircraft. Roughness effects on transition are also studied for blunt re-entry capsules (Fig. 12) as part of a DFG project together with several German universities. Furthermore, centrifugal and Coriolis force terms were recently added to the instability and transition analysis tool NOLOT/PSE which is now used for transition studies in rotating frames of reference like the boundary-layer flow on blades of helicopters, wind turbines, and propellers. The numerical work is complemented by detailed experimental studies on disturbance development in transitional flows and transition location measurements on different configurations. As an example, the High-Speed Infra-Red (HSIR) thermography method has successfully been applied in different industrial low-speed facilities (Airbus LSWT in Bremen, RUAG LWTE in Emmen (CH)) to determine transition locations on propeller blades rotating with speeds of up to 14500 RPM. In high-speed flows, experimental investigations of the 2-D impinging shock-wave interaction with the transitional flat-plate boundary layer were conducted in the Ludwieg-Tube Facility at DLR Göttingen at Mach 6 flow conditions. The highest surface-heating loads were detected in cases when the shock waves hit the boundary layer just behind the transition onset location.

More information:
`Andreas.Krumbein@dlr.de`, `Stefan.Hein@dlr.de`

5 Contact Address

Prof. Dr.-Ing. M. Sommerfeld
Otto-von-Guericke Universit at Magdeburg
Institut für Verfahrenstechnik
AG Mehrphasenströmungen
Zeppelinstraße 1
D-06130 Halle (Saale), Germany

Tel.: (0049)-345-5523680
Fax: (0049)-345-5527586
E-mail: `martin.sommerfeld@ovgu.de`
homepage: `http://www.mps.ovgu.de`