Chair of Hydromechanics
LEHRSTUHL FÜR HYDROMECHANIK
PROF. DR.-ING. MICHAEL MANHART

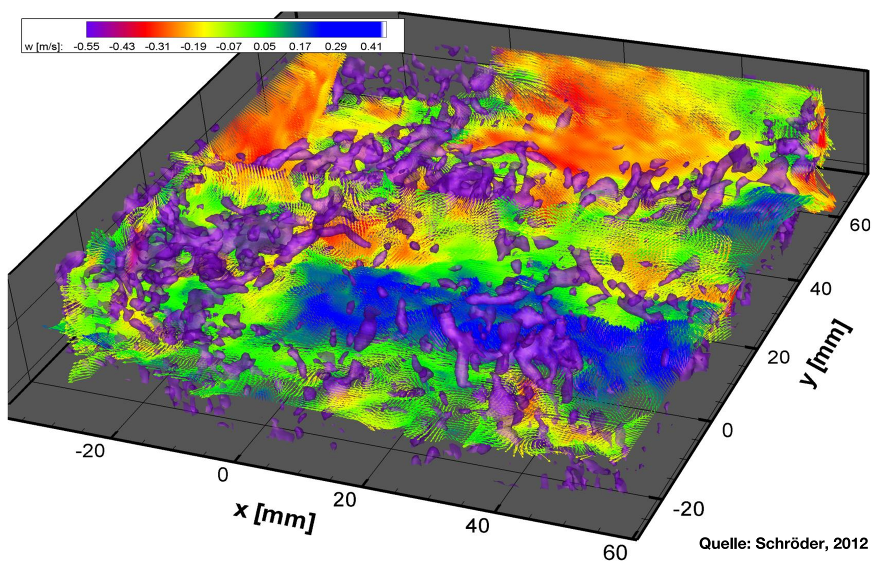
RESPONSIBILITIES IN EDUCATION AT THE CHAIR OF HYDROMECHANICS

The Chair of Hydromechanics is responsible for the education in Hydromechanics which is one of the classical fundamental fields in Civil and Environmental Engineering. The teaching ranges from the fundamental lecture in "Hydromechanics" (Bachelor-level) to specialized courses in turbulence modeling and numerical methods. Another important aspect is the training of the students in applied and scientific methods for flow measurements which takes place in the chair's laboratory. Courses are given for Bachelor's and Master's education in Civil and Environmental Engineering, for the Master's education in Computational Mechanics and for Bachelor in Engineering Sciences.

THE LABORATORY – FROM EDUCATION TO RESEARCH

With a lab space of 1600 square meters, the Hydromechanics Laboratory spans the bridge from applied hydraulic research to fundamental research in fluid mechanics. Starting from 2004 modern laser-based equipment has been acquired to enable time- and space-resolved measurements of turbulent flow quantities. With its capability for large-scale hydraulic setups the lab serves as a platform for consulting of partners from industry and government authorities; results from DFG-funded fundamental research find their way into the international research community by publications and as reference data in data bases (ERCOFTAC); a variety of hands-on experiments assist teaching and let the students gain their own experiences in special designed experimental setups.

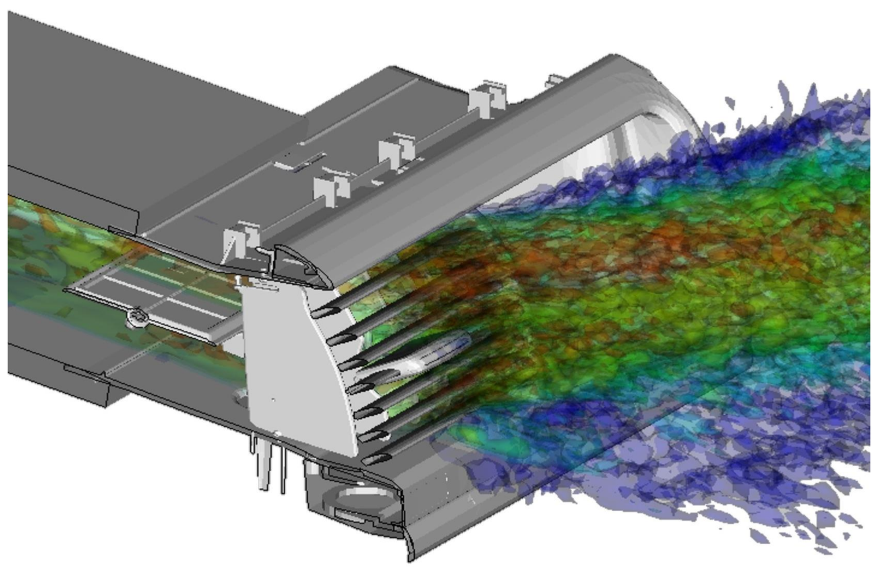
Teaching is one of the laboratory's main tasks. Doing experiments supervised or self dependent, students can achieve basic understanding of hydromechanic effects, (Ernst Otto Fischer-Lehrpreis, 2011).



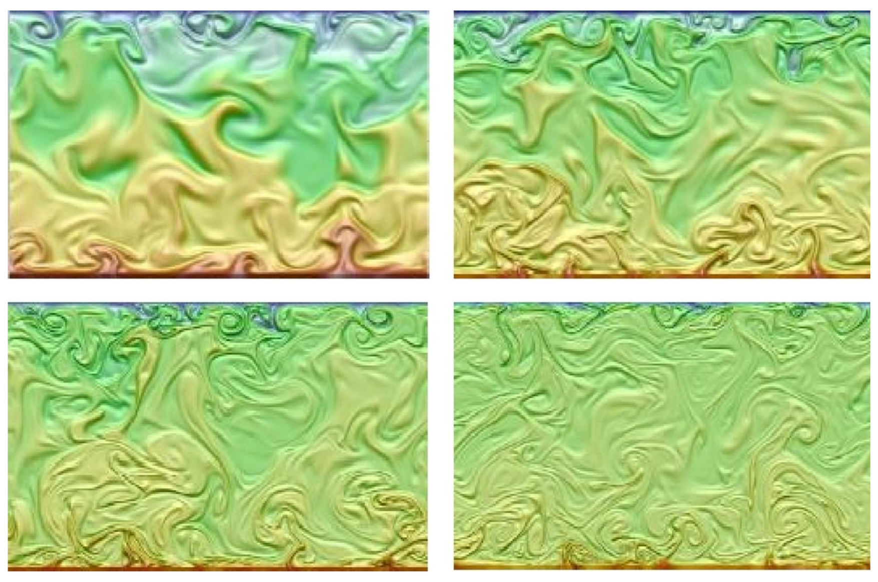
Visualisation of a part of an instantaneous flow field over periodic hills in the Chair's lab. Tomographic 3D Particle Image Velocimetry, taken during a measurement campaign for validation of Tomographic PIV by DLR Göttingen. (Schröder et al., 2012).

MGLET – FROM NUMERICS TO PHYSICS

The in-house code MGLET is a Finite Volume solver specially designed for large-scale space- and time-resolved simulations of turbulent flows. It is being used for fundamental research by Direct Numerical Simulations (DNS) and Large Eddy Simulations (LES) of complex turbulent flows on current high performance computers (more than 10^9 grid cells). It is being used by other partners (e.g. NTNU Trondheim, DLR Institute of atmosphere physics) with numerous publications in highly rated scientific journals. Recent developments include (i) conservative Immersed Boundary method for arbitrary geometries in a Cartesian grid (ii) zonal grid refinement and hierarchical grids for high Schmidt number scalar fields (iii) particle dynamics and fiber suspension (iv) fourth-order Finite Volume schemes.



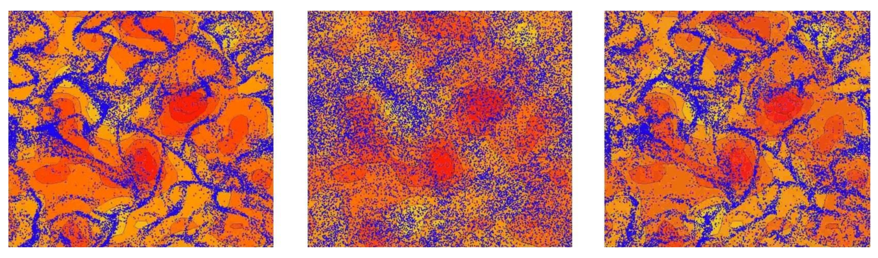
Investigation of noise generation in an HVAC system inside of a car, by LES with the Chair's in house code MGLET. The noise sources (according to Lighthill's analogy) are located in regions with high shear and turbulent kinetic energy, (LES by MGLET; Kreuzinger, 2009).



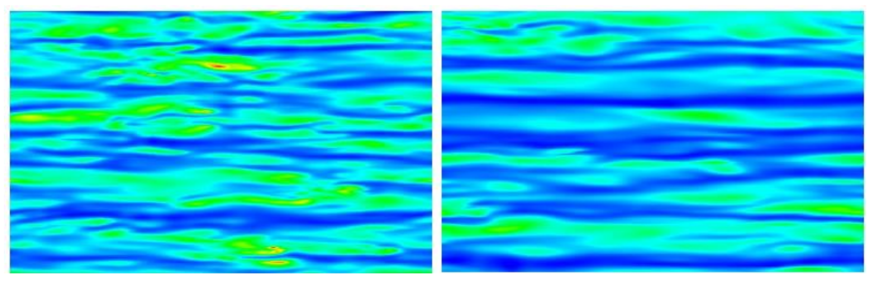
Scalar fields in turbulent channel flow at different Schmidt numbers, increasing from three (above, left) to 49 (lower, right). One can see that scalar concentration fluctuations are of increasingly finer structure with increasing Schmidt number (decreasing diffusivity), DNS by MGLET (Schwertfirm & Manhart, Int. J. Heat and Fluid Flow, 2007; Da Vinci Award 2009 by ERCOFTAC).

MULTIPHASE FLOWS – FROM PARTICLES TO FIBERS

Particle-laden flows are investigated both by Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES). The particles are tracked by a Lagrangian tracking with a conservative interpolation scheme. We quantified the impact of small scale effects on the particles by DNS of isotropic turbulence and developed a model for subgrid scale effects on the particles in LES.
We developed a two-way coupled method for dilute suspensions of rigid fibers in turbulent flows. This method uses a microscopic Monte-Carlo method to describe the dynamics of the fibers in orientation space. This development helped us to study turbulent drag reduction by rigid fibers and developed macroscopic closure models for turbulent fiber flows.



Instantaneous paticle distribution in an isotropic turbulent flow field for different subgrid scale models. DNS (left); stochastic method from literature (middle); new model based on spectrally optimised interpolation kernels (right), (DNS and LES by MGLET, Gobert & Manhart, J. of Computational Physics, 2011).



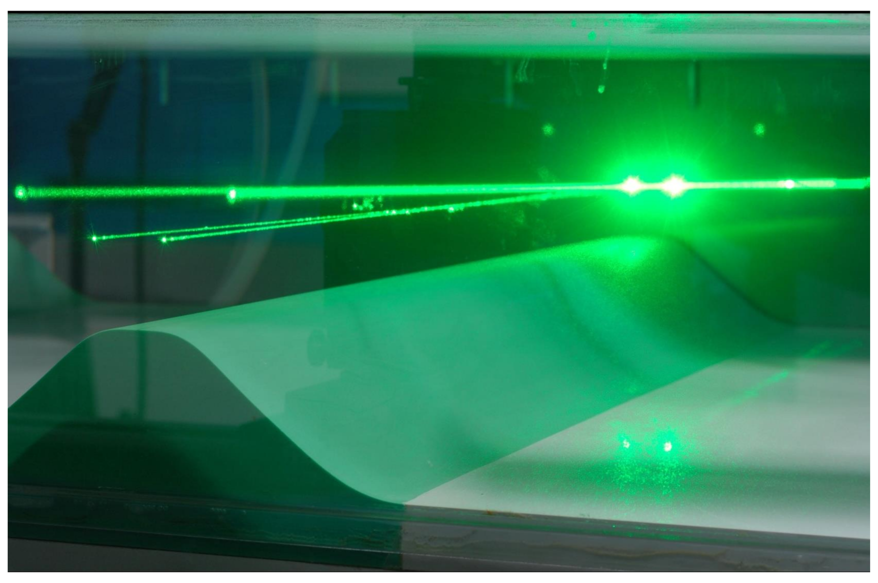
Fluctuations of the streamwise velocity component in a wall-parallel plane in a turbulent channel flow. Newtonian flow (left) and fiber suspension (right). The fibers increase anisotropy in the flow and thus can reduce the overall drag, (DNS by MGLET; Moosaie & Manhart, submitted to Acta Mechanica, 2012).

FUNDAMENTAL AND APPLIED RESEARCH AT THE CHAIR OF HYDROMECHANICS

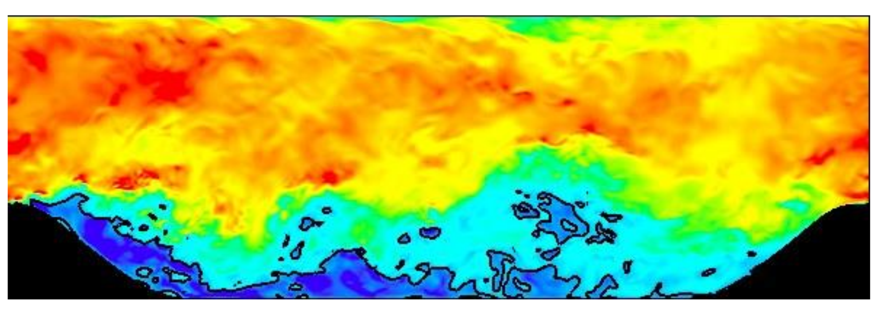
The research ranges from fundamental research on turbulent flows to applied hydraulic research. The field of porous media flows is relatively new for the Chair. The research on turbulent flows profits from a long-term development of the simulation software MGLET - a code which has been developed for the detailed and time-resolved simulation (Large-Eddy and Direct Numerical Simulation) of turbulent flows. This code is used for development of new models for geometrically complex flows and for two-phase flows with suspended particles or fibers, and as a basis for the investigation of detailed (microscopic) processes in such flows. The code development and validation is closely linked to experimental measurements in the Laboratory of the Chair. In such a way, one of the strengths of the Chairs is the combination of numerical methods with experimental methods. The applied research in the field of Hydraulics is an important factor for establishing the bridge between university and practice which is especially important and useful for a practical education of the students.

COMPLEX TURBULENT FLOWS – FROM EXPERIMENTAL TO NUMERICAL RESEARCH

The flow over periodically arranged 2D hills serves as a prototype for flows with separation and reattachment showing a variety of non-linear flow phenomena that can not be treated by standard Reynolds Averaged methods (such as k-ε models). Originally having been used as a Benchmark in the French-German research group on 'LES of Complex Flows', a validation experiment has been built in the Chair's lab. The flow field was measured by Laser-Doppler-Anemometry and Particle Image Velocimetry in selected profiles and planes. The results found their way into ERCOFTAC's reference data base. The setup is being used to explore near wall effects close to the hill top which have been identified as the main source of modeling errors. Other research partners have been testing their equipment recently in this channel, e.g. 3D time-resolved tomographic PIV.



Laser-Doppler-Anemometry of the flow over periodic hills in the Chair's Laboratory, (Rapp & Manhart, Exp. Fluids, 2011).

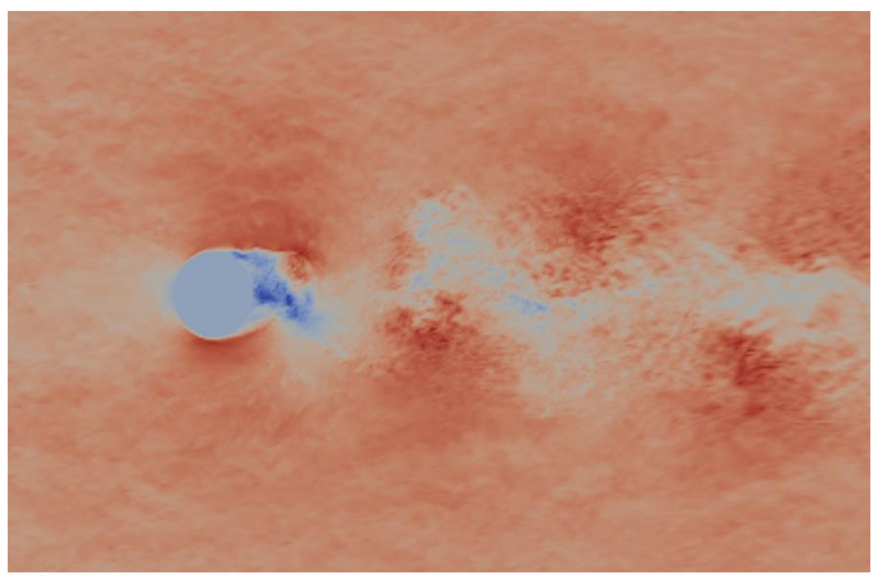


Direct Numerical Simulation of the flow over periodic hills with the Chair's in house code MGLET, (Manhart et al., Theor. Comput. Fluid. Dyn., 2008).

LOCAL SCOUR AT A BRIDGE PIER – FROM MICROSCOPIC TO ENGINEERING MODELS

This research field emphasises the benefit of combined numerical and experimental studies. To gain a deeper insight into scour development in the vincinity of a cylindrical bridge pier, extensive experiments have been conducted in the Chair's Laboratory. The results allow the analysis of the erosion processes and their evolution during scour hole development. The next step aims at developing numerical methods for scour prediction around such a bridge pier. The main strategy is linking Large Eddy Simulation for the flow field to a model for sediment transport and soil deformation. The LES results will be validated by data fom the experiment.

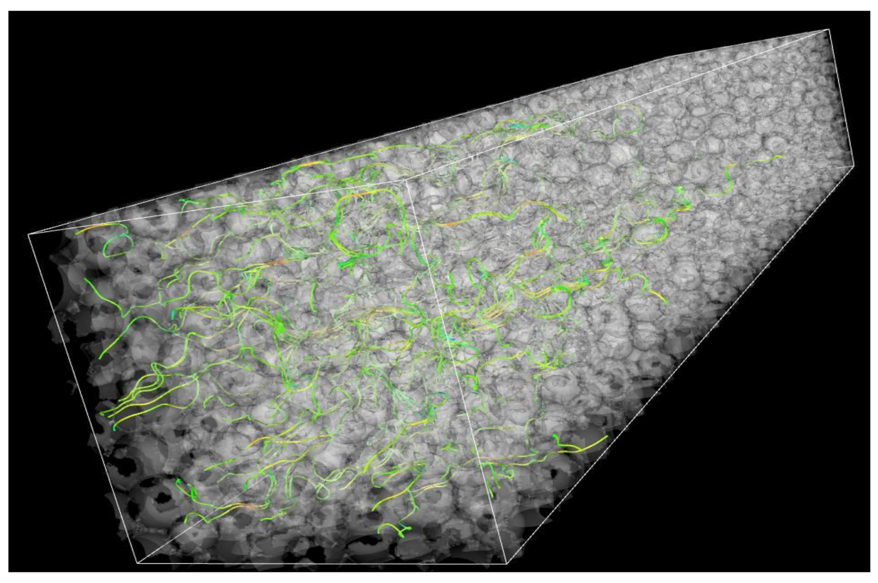
Scour hole around a cylinder in a sandy bed. Photograph taken during the experiments in the Chair's Laboratory. (Pfleger et al., Int. Conf. Scour & Erosion, 2011).



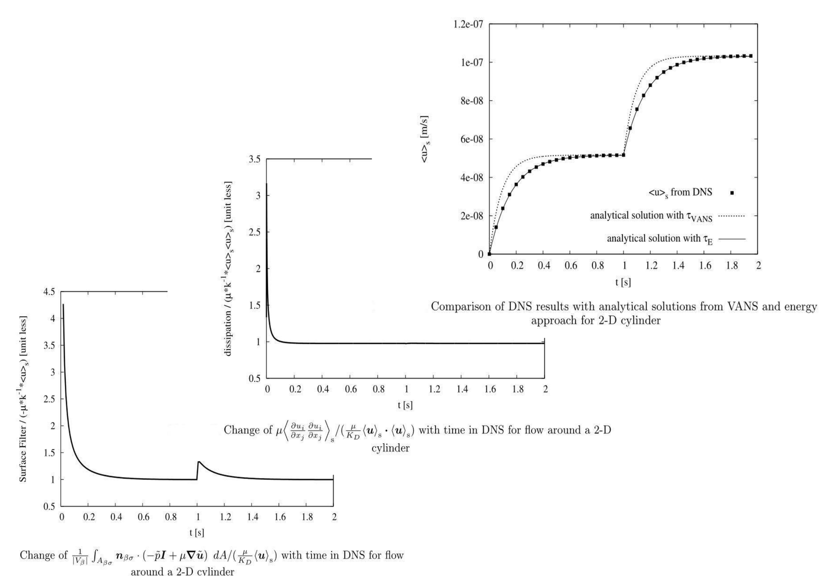
Velocity field (u-component) of the turbulent flow around a cylinder mounted on a flat plate. LES by MGLET (Schanderl, 2012).

POROUS MEDIA FLOWS – FROM MICRO- TO MACROSCALE

Porous media flows are a new field of research at the Chair of Hydromechanics. We concentrate on the study of pore scale effects and their modeling on the macroscopic scale. These effects include (i) dispersion, (ii) time scale in unsteady flow through a porous medium and (iii) interaction of a turbulent flow with a porous medium. By performing numerical simulations of the flow in the pore space (DNS) we are able to determine the macroscopic parameters without any modeling. The simulations have been validated by careful grid studies and empirical correlations for sphere packs.



Streamlines in a porous media flow, modelled by a random sphere pack. DNS by MGLET (Kooshapur & Manhart, Simtech, 2011).



Time-dependent flow through a porous medium after sudden changes of pressure gradient. Validation of a new closure for the time scale in unsteady Darcy's equation by DNS. Zhu et al. (in preparation).