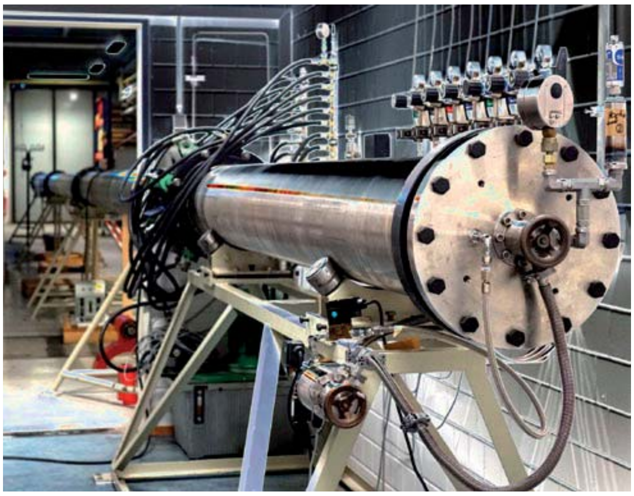
Aerodynamics and Fluid Mechanics

Numerical modeling, simulation and experimental analysis of fluids and fluid flows

The focus of the Institute of Aerodynamics and Fluid Mechanics in 2016-17 was on further development of a multi-resolution parallel simulation environment for the NANOSHOCK project, on reduced-order modeling of fluid-structure interaction, on the analysis of advanced aerodynamic configurations for helicopter, aircraft and automobiles, and on advanced simulation and gridding technologies for exterior and interior aerodynamics.

A highlight in 2016-17 was the successful operation start of the large shock-tube facility, and the kick-off of two interdisciplinary DFG projects with the institutes IWB and FZG, where the institute brings in its expertise on advanced flow-simulation methodology. Dr. Lin Fu, graduating from the institute in 2017, received a Postdoctoral Fellowship from Stanford University, and M.Sc. Thomas Paula received the Willy Messerschmitt award for his Master Thesis absolved at the German Association for Aero- and Astronautics. Last but not least, from the NANOSHOCK project the first CFD code spin-off was opened for perusal by the scientific community: https://www.aer.mw.tum.de/abteilungen/nanoshock/news

Experimental and Numerical Investigation of Cavitating Two-phase Flows and Cavitation-induced Erosion



Shock tube at AER – length 24 m, diameter 0.3 m, pressures from 1 Pa to 50 bar

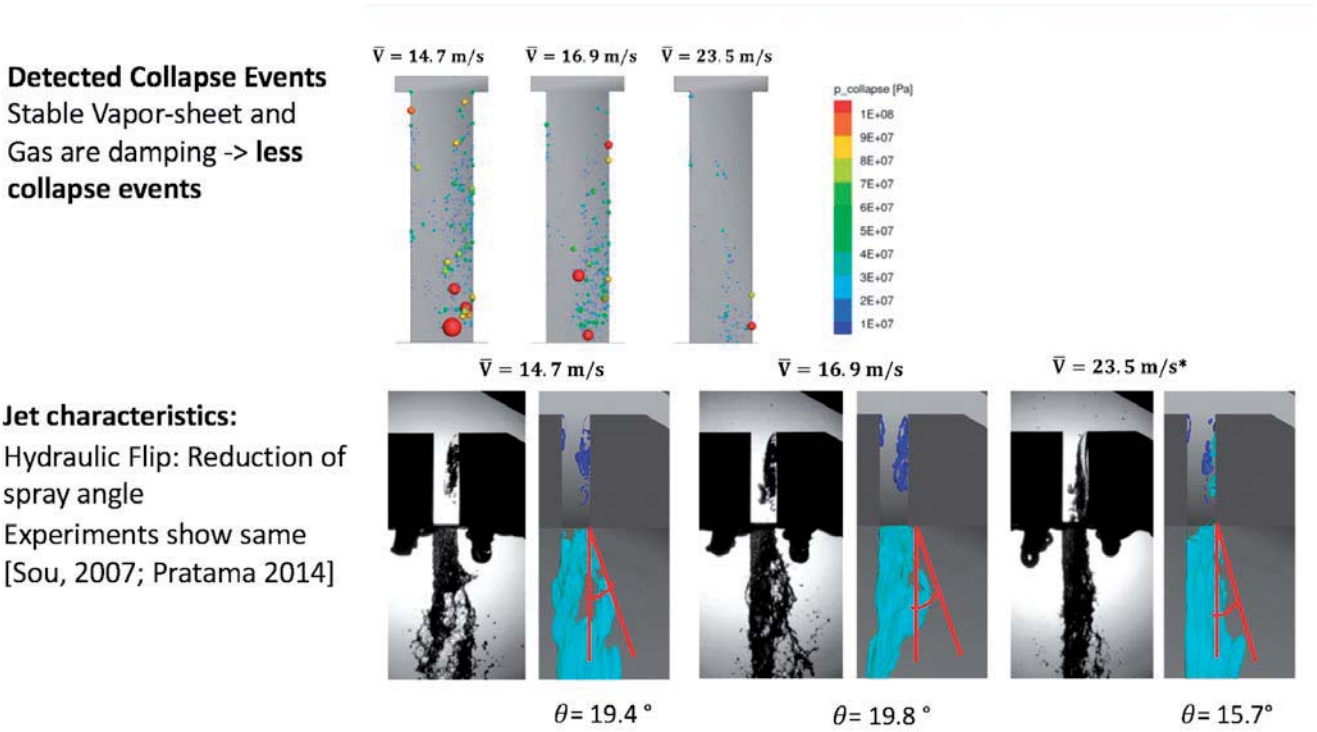
Motivation and Objectives

The formation of vapor bubbles in a liquid due to pressure reduction is called 'cavitation'. Flows involving cavitation feature a series of unique physical properties such as discontinuous jumps in the speed of sound from O(1000) m/s to O(1) m/s, a jump in density of up to four orders of magnitude, and intense compressibility effects, such as the formation of intense shock waves with post-shock pressures of more than 1GPa. Flows involving cavitation occur in a wide range of technical systems. In particular, injection systems for combustion engines, high pressure hydraulics, naval propellers and biomedical applications are prone to cavitation and cavitation-induced material erosion.

Our objective is to develop efficient and accurate simulation approaches for predicting all dominating phenomena in cavitating flows including shock-wave formation and propagation, with the goal to provide the groundwork for the design optimization of future technical devices.

Approach to Solution

We perform fundamental experiments using a shock tube and state-of-the art high speed cameras/sensors to investigate collapse processes of gas und vapor bubbles embedded in a liquid-like gel. These experiments are used to enhance physical understanding of involved fluid dynamics and serve as reference data to our numerical investigations. For about one decade, mathematical models and numerical approaches for efficient and accurate predictions of cavitating flow phenomena have been developed at the institute. A series of numerical approaches, including state-of-the-art large-eddy simulation (LES) schemes enable high performance computing with linear scaling on HPC systems, such as SuperMUC. Our approaches are 'monolithic' in a sense that all fluid components involved (liquid, vapor, inert gases) are handled in a consistent way. Shock-wave formation due to collapsing vapor patterns is resolved by application of time steps smaller than one nanosecond. The resulting loads on material surfaces – and thus the potential of material erosion – are obtained without the need for additional models. Fundamental research is funded by the European Union (Project 'CaFE'), while applied research is performed in collaboration with several automotive suppliers, the U.S. Office of Naval Research and with the European Space Agency.



Spray angle and erosion analysis in generic injector components.

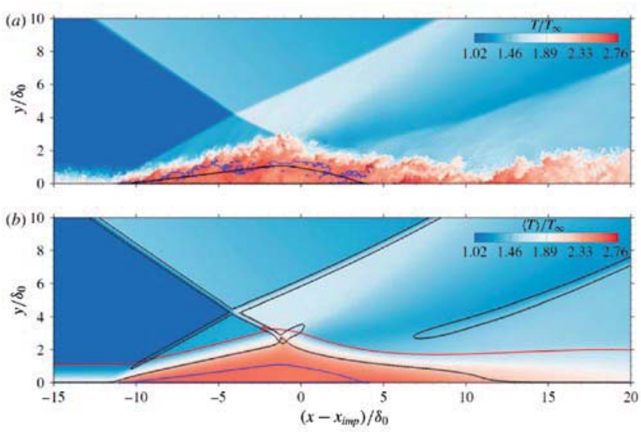
Key Results

- Budich, B.; Schmidt, S.J.; Adams, N.A.: Numerical Simulation and Analysis of Condensation Shocks in Cavitating Flow. To appear in Journal of Fluid Mechanics (accepted 2017)
- Beban, B.; Schmidt, S.J.; Adams, N.A.: Numerical study of submerged cavitating throttle flows. Atomization and Sprays (27 (8)), 2017, pp. 723-739
- Egerer, C.P.; Schmidt, S.J.; Hickel, S.; Adams, N.A.: Efficient implicit LES method for the simulation of turbulent cavitating flows. Journal of Computational Physics, Volume 316, 2016, pp. 453-469
- Örley, F.; Hickel, S.; Schmidt, S.J.; Adams, N.A.: Large-Eddy Simulation of turbulent, cavitating fuel flow inside a 9-hole Diesel injector including needle movement. International Journal of Engine Research, 2016, pp. 1-17

Numerical Methods for Computational Fluid Dynamics
Physical Consistency of High-resolution CFD

Motivation and Objectives

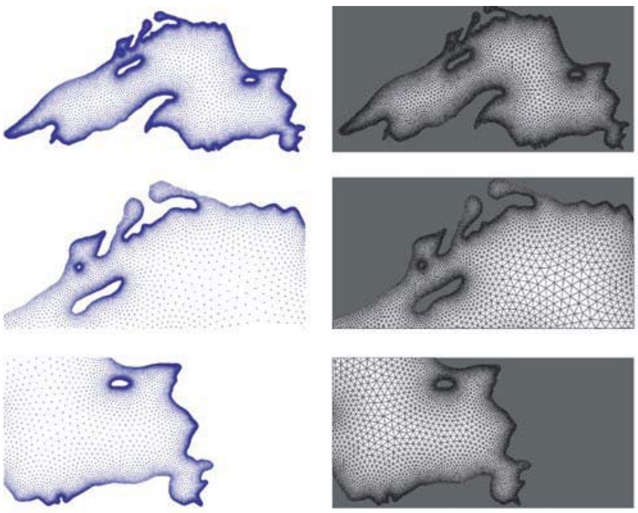
CFD tools in physical or engineering applications can never reach numerical resolution levels where the truncation error of the discretization schemes enters its asymptotic limit. It thus is of high practical relevance to design schemes that have good scale resolution properties whenever numerical resolution is sufficient for relevant flow scales, and whose truncation error functions as a physically consistent subgrid-scale model when not. In the past, this research concept has led to the development of the first physically consistent and practically successful implicit LES model. Currently, the notion of employing model uncertainty and truncation errors as physical-model surrogates is being pursued on several levels.



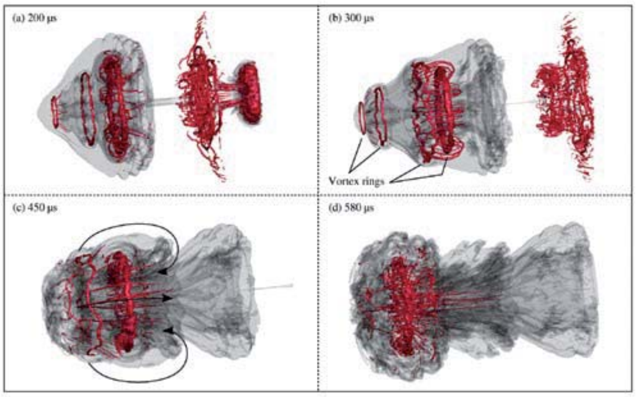
Instantaneous contours of temperature (top) of Ma=3 shock impinging on a turbulent boundary layer, and corresponding mean flow (bottom)

Approach to Solution

Concepts of physical design of modeling and discretization error have been successfully employed for further development of high-resolution schemes and targeted ENO schemes that are suitable for underresolved computations of turbulent and non-turbulent flows. The physically consistent implicit LES model ALDM has been applied to turbulent shock-boundary-layer interaction at unprecedented Reynolds numbers. Extending the general concept to numerical models for fluctuating hydrodynamics, the manipulation of modeling errors within the dissipative-particle-dynamics model is investigated to explore spontaneous long-range correlations in turbulent flows. Physical effects of truncation errors in particle-discretizations may also lead to relaxation processes that allow for highly effective mesh generation and domain partitioning methods. Both have been developed to a pre-commercialization demonstration level.



Automatic generation of unstructured meshes with high mesh quality based on physical-particle advection anaology



Vortex-ring evolution for reacting shock-bubble interaction, grey shades correspond to inert-gas mass fraction and red is isosurface of vorticity v

Key Results

- Pasquariello, V.; Hickel, S.; Adams, N.A.: Unsteady effects of strong shock-wave/boundary-layer interaction at high Reynolds number. Journal of Fluid Mechanics (823), 2017, pp. 617-657 mehr … BibTeX Volltext (DOI)
- Diegelmann, F., Hickel, S., Adams, N.A.: Three-dimensional reacting shock–bubble interaction. Combustion and Flame 181, 2017, pp. 1339-1351 mehr … BibTeX Volltext (DOI)
- Fu, L.; Hu, X.Y.; Adams, N.A.: Targeted ENO schemes with tailored resolution property for hyperbolic conservation laws. Journal of Computational Physics (349), 2017, pp. 97-121 mehr … BibTeX Volltext (DOI)
- Fu, L.; Hu, X.Y.; Adams, N.A.: A physics-motivated Centroidal Voronoi Particle domain decomposition method, Journal of Computational Physics 335, 2017, pp. 718-735 mehr … BibTeX Volltext (DOI)
- European Patent Application EP32556611, Hu X., Fu L., Han L., Adams N.A., Method and system for generating a mesh

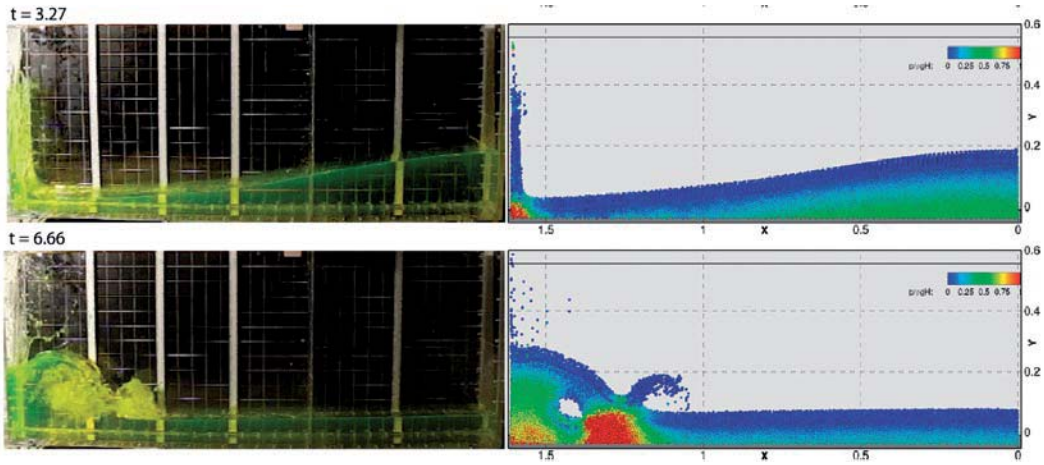
Smoothed Particle Hydrodynamics Method for Simulating Free Surface Flow

Motivation and Objectives

Smoothed particle hydrodynamics (SPH) is a purely mesh-free Lagrangian method developed for astrophysical applications. Since these pioneering works, the SPH method has been successfully applied for numerical simulations of solid mechanics, fluid dynamics and fluid-structure interaction. Concerning the computation of hydrodynamic problems, the present methods either lead to violent pressure oscillations or excessive dissipation and are not able to reproduce correct physical phenomena reliably.

Approach to Solution

We present a low-dissipation, weakly-compressible SPH method for modeling free-surface flows exhibiting violent events such as impact and breaking. The key idea is to modify a Riemann solver which determines the interaction between particles by using a simple limiter to decrease the intrinsic numerical dissipation. The modified Riemann solver is also extended for imposing wall boundary conditions. Numerical tests show that the method resolves free-surface flows accurately and produces smooth, accurate pressure fields.



Three-dimensional dam-break problem simulated with dp =H/30 (the total fluid particle number N=27000): free-surface profile compared with experiment.

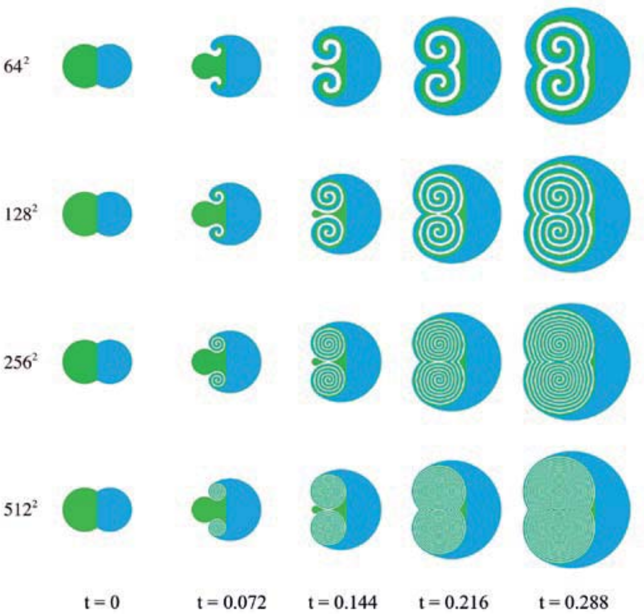
Key Results

- A weakly compressible SPH method based on a low-dissipation Riemann solver, C. Zhang, X.Y. Hu, N.A. Adams. Journal of Computational Physics 337 (2017) 216–232.
- A generalized transport-velocity formulation for smoothed particle hydrodynamics, C. Zhang, X.Y. Hu, N.A. Adams. Journal of Computational Physics 337 (2017) 216–232.
- Targeted ENO schemes with tailored resolution property for hyperbolic conservation laws, L. Fu, X.Y. Hu, N.A. Adams, Journal of Computational Physics 349 (2017) 97–121.

Numerical Modeling of Interface Networks

Motivation and Objectives

Multi-region problems can occur when the motion of more than two immiscible fluids is to be described. In this case the interface network, separating the different fluid regions, evolves in time due to interactions of the different fluids across interface segments. These interactions can often be described by local fluid properties. Due to the complexity of the topology, numerical modeling of the evolution and interactions near the interface network are long-standing challenges for the research community



Constant normal driven flow of an interface network with three regions at different time instance

Approach to Solution

We have developed a high-resolution transport formulation of the regional level-set approach for an improved prediction of the evolution of complex interface networks. The approach thus offers a viable alternative to previous interface-network level-set method.

Key Results

- High-resolution method for evolving complex interface networks, S.C. Pan, X.Y. Hu, N.A. Adams, Computer Physics Communication, accepted for publication 2017
- High-order time-marching re-initialization for regional level-set functions. S.C. Pan, X.X. Lyu, X.Y. Hu N.A. Adams, Journal of Computational Physics, accepted for publication 2017
- Single-step re-initialization and extending algorithms for level-setbased multi-phase flow simulations. L. Fu, X.Y. Hu, N.A. Adams, accepted for publication 2017

Aircraft and Helicopter Aerodynamics

Model of BLUECOPTER configuration mounted in the test section of wind tunnel A.

Motivation and Objectives

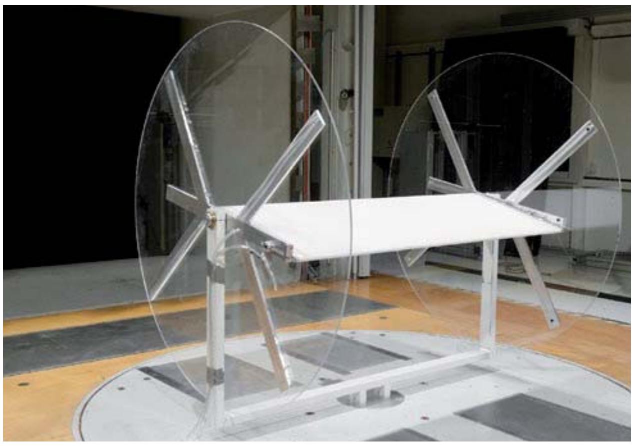
The long-term research agenda is based on the continued improvement of flow simulation and analysis capabilities in the context of aircraft and helicopter performance enhancement and drag reduction. Specific research activities are dedicated to the reliable prediction of flow separation onset and progression in the context of vortex dominated flow and control of leading edge vortex systems, development of a novel ROM framework for aeroelastic analysis, helicopter drag reduction of rotor hub and engine intake by shape optimization and flow control, development of propeller performance and optimization tool chain with respect to electrically driven flight vehicles and fluid-structure interaction of membrane-type lifting surfaces applied to wind turbine rotors.

Approach to Solution

The investigations have been performed using both wind tunnel experiments and state-of-the art numerical simulations. In-house codes are continously elaborated further in the context of aeroelasticity analysis with respect to time-accurate, fully-coupled simulations as well as the application of novel neuro-fuzzy based reduced order models. Commercial CFD codes are applied to flow control problems and helicopter aerodynamics addressing unsteady loads analysis and aeroacoustics.

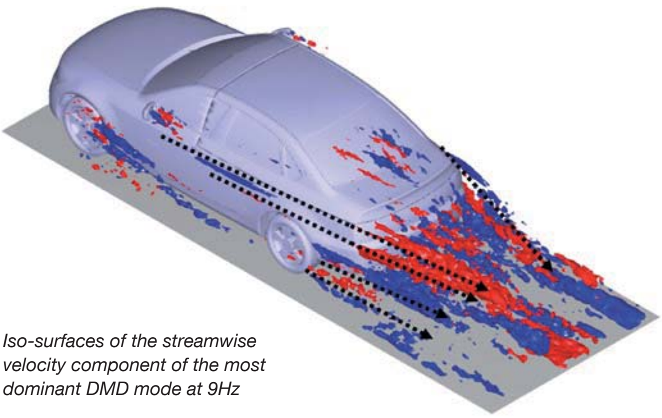
Key Results

- Buzica, A., Bartasevicius, J. and Breitsamter, C.: Experimental investigation of high-incidence delta-wing flow control. Experiments in Fluids, Vol. 58:131, 2017
- Buzica, A. and Breitsamter, C.: Experimental and Numerical Investigation on Delta-Wing Post-stall Flow Control. NNFM, Vol. 136, Springer 2017, pp. 167-177
- Knoth, F. and Breitsamter, C.: Flow Analysis of a Helicopter Engine Side Air Intake. Journal of Propulsion and Power, Vol. 33, No. 5, 2017, pp. 1230-1244
- Knoth, F. and Breitsamter, C.: Aerodynamic Characteristics of Helicopter Engine Side Air Intakes. Aircraft Engineering and Aerospace Technology, 2017
- Knoth, F. and Breitsamter, C.: Aerodynamic Testing of Helicopter Side Intake Retrofit Modifications. Aerospace, Vol. 4, 33, 2017, pp. 1-17
- Knoth, F. and Breitsamter, C.: Numerical and Experimental Investigation of a Helicopter Engine Side Intake. NNFM, Vol. 136, Springer, 2017, pp. 27-39
- Piquee, J. and Breitsamter, C.: Numerical and Experimental Investigations of an Elasto-Flexible Membrane Wing at a Reynolds Number of 280.000. Aerospace, Vol. 4, 39, 2017, pp. 1-18
- Piquee, J., Saeedi, M., Breitsamter, C., Wüchner, R. and Bletzinger, K.-U.: Numerical Investigation of an Elasto-Flexible Membrane Airfoil Compared to Experiments. NNFM, Vol. 136, Springer, 2017, pp. 421-431
- Rozov, V., Hermanutz, A., Breitsamter, C. and Hornung, M.: Aeroelastic Analysis of a Flutter Demonstrator with a very Flexible High-Aspect-Ratio Swept Wing. IFASD-2017-173, 2017
- Winter, M., Heckmeier, F. and Breitsamter, C.: CFD-Based Aeroelastic Reduced-Order Modeling Robust to Structural Parameter Variations. Aerospace Science and Technology, Vol. 67, 2017, pp. 13-30
- Winter, M. and Breitsamter, C.: Coupling of Recurrent and Static Neural Network Approaches for Improved Multi-step Ahead Time Series Prediction. NNFM, Vol. 136, Springer, 2017, pp. 433-442
- Winter, M. and Breitsamter, C.: Application of Unsteady Aerodynamic Reduced-Order Modeling Techniques to a Complex Configuration. IFASD-2017-217, 2017



Test bed of an elasto-flexible membrane wing configuration mounted in the test section of wind tunnel A

Modal Decomposition for Vehicle Aerodynamics



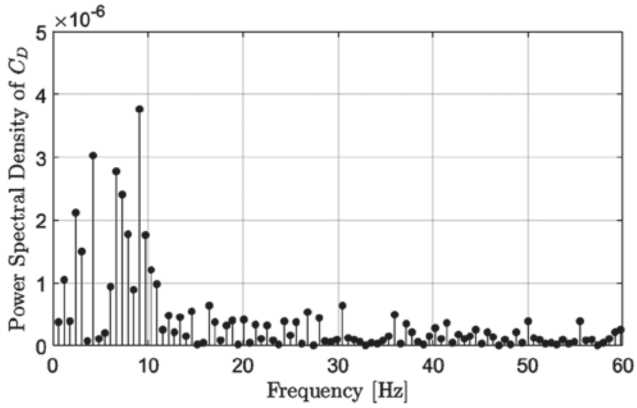
Iso-surfaces of the streamwise velocity component of the most dominant DMD mode at 9Hz

A new modal decomposition approach is employed for analyzing temporally resolved flow field data from detached eddy simulation (DES) using the open source CFD environment OpenFOAM®. Velocity components are temporally filtered with moving average filter before being interpolated to a coarser equidistant mapping mesh. These filtering operations reduce the amount of spurious numerical oscillations in the data to be analyzed and cut off high frequency, low energy content. In order to extract the most dominant flow structures for in-depth analysis, an incremental variant of dynamic mode decomposition (DMD) was found to be most useful. DMD generates modes of distinct frequency that can be reconstructed in time. Several modifications to an already existing variant are implemented to increase the applicability for large data sets, mainly reducing the required amount of memory, which is the most limiting factor in modal analysis for industrial applications with large data sets. The modes represent flow structures of vortex shedding and stationary recirculation processes. Reconstruction enables tracking of structures to their respective excitation mechanisms and allows for identification of geometrical features that introduce strong perturbations to the flow. Strong perturbations lead to an increase in potential for viscous dissipation in the wake of bluff bodies and thus to generally lower base pressure and increased drag.

The DrivAer reference model in notchback configuration with structured underbody, engine bay flow, open wheel houses and open rotating rims is simulated in wind tunnel conditions with a rolling road system. The results from CFD are processed using the DMD approach described above and the most dominant flow structures are visualized and discussed. Small scale detachments that are generated far upstream of the mean detachment line of the vehicle's rear end travel downstream along the surface, triggering large-scale structures in the wake. The frequency of those structures is also dominant in the frequency spectra of the integrated force coefficient.

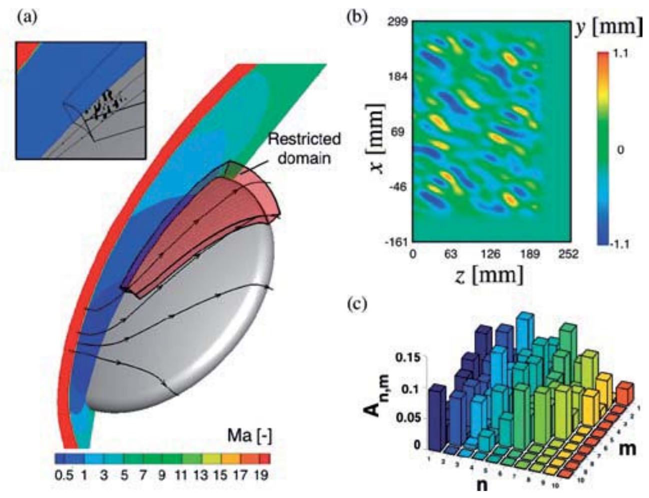
Publications

- Kiewat, Haag, Indinger, Zander, 'Low-Memory Reduced-Order Modelling with Dynamic Mode Decomposition Applied on Unsteady Wheel Aerodynamics', 2017 ASME Fluids Engineering Division Summer Meeting
- Kiewat, Haag, Matsumoto, Indinger, Zander, 'Online Dynamic Mode Decomposition for the Investigation of Unsteady Vehicle Aerodynamics of the DrivAer Model: Part 2. Application on Velocity Fields', 2017 JSAE Annual Congress (Spring)



Frequency spectrum of the integrated drag force coefficient

Laminar-turbulent Transition with Chemical (Non-)Equilibrium in Hypersonic Boundary-Layer Flows



Computational setup for a roughness patch on a re-entry capsule

Motivation

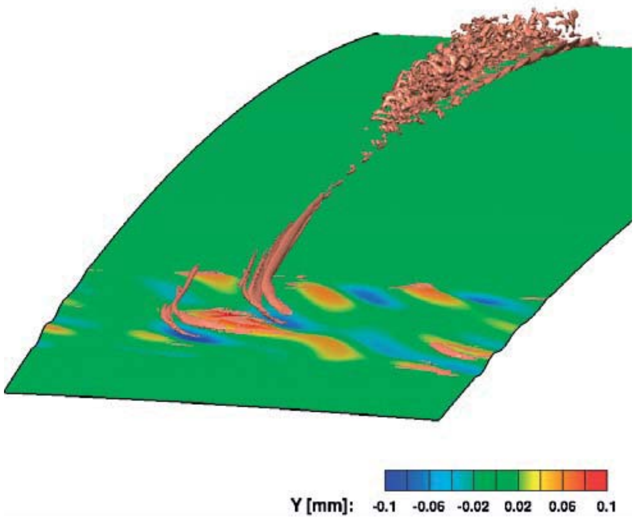
Blunt bodies returning from space are subject to immense heat loads leading to ablation. Roughnesses on these ablating surfaces can induce laminar-turbulent transition in an otherwise laminar flow. Laminar-turbulent transition increases the heat load on the surface. This self-energizing effect can lead to a catastrophic failure of the spacecraft. The role of the chemical modelling in high-temperature boundary layers in equilibrium and non-equilibrium is the main focus of the numerical work.

Approach to Solution

Direct numerical simulations (DNS) are conducted on national HPC facilities such as SuperMUC and HLRS. Results show that roughness wakes are subject to an increased instability in the presence of chemical reactions and non-equilibrium effects

Key Results

- A. Di Giovanni and C. Stemmer: Numerical Simulations of the High-Enthalpy Boundary Layer on a Generic Capsule Geometry with Roughness. New Results in Numerical and Experimental Fluid Mechanics XI (Notes on Numerical Fluid Mechanics and Multidisciplinary Design 136), 2017. Contributions to the 20th STAB/ DGLR Symposium Braunschweig, Germany, 2016, pp. 189-199
- C. Stemmer, M. Birrer and N.A. Adams: Disturbance Development in an Obstacle Wake in a Reacting Hypersonic Boundary Layer. Journal of Spacecraft and Rockets (54(4)), 2017, pp. 945-960 and pp. 899-915



DNS results from vortical disturbances induced by surface roughness in a reacting environment. The wake of the roughness elements becomes unstable and laminar-turbulent transition takes place.

Aerodynamics and Fluid Mechanics

Prof. Dr.-Ing. Nikolaus A. Adams

Contact
www.aer.mw.tum.de
nikolaus.adams@tum.de
Phone +49.89.289.16138

Management
Prof. Dr.-Ing. Nikolaus A. Adams, Director
Apl. Prof. Dr.-Ing. habil. Christian Breitsamter
PD Dr.-Ing. habil. Christian Stemmer
Dr.-Ing. Xiangyu Hu
PD Dr.-Ing. habil. Thomas Indinger

Adjunct Professors
Prof. i.R. Dr.-Ing. habil. Rainer Friedrich
Prof. em. Dr.-Ing. Boris Laschka, Emeritus
Apl. Prof. i.R. Dr.-Ing. Hans Wengle, Emeritus

Visiting Lecturer
Dr.-Ing. Rainer Demuth (BMW Group)

Administrative Staff
Angela Grygier
Amely Schwörer
Li Su, M.Sc.
Dip.-Betriebw. (FH) Sandra Greil

Research Staff
Dr.-Ing. Stefan Adami
Dr. -Ing. Xin Bian
Vladimir Bogdanov, M.Sc.
Morgane Borreguero, M.Sc.
Dipl.-Ing. Andrei Buzica
Michael Cerny, M.Sc.
Antonio Di Giovanni, M.Sc.
Alexander Döhring, M.Sc.
Zheng Fang, M.Sc.
Nico Fleischmann, M.Sc.
Lin Fu, M.Sc.
Dr.-Ing. Marcus Giglmaier
Polina Gorkh, M.Sc.
Dipl.-Ing. Lukas Haag
Dr.-Ing. Luhui Han
Florian Heckmeier, M.Sc.
Thomas Hopfes, M.Sc.
Nils Hoppe, M.Sc.
Naeimeh Hosseini, M.Sc.
Xiaobin Hu, M.Sc.
Zhe Ji, M.Sc.
Jakob Kaiser, M.Sc.
Dipl.-Ing. Thomas Kaller
Marco Kiewat, M.Sc.
Sebastian Klukas, M.Sc.
Dipl.-Ing. Florian Knoth
Andreas Kümmel, M.Sc.
Christian Lang, M.Sc.
Yue Li, M.Sc.
Aleksandr Lunkov, M.Sc.
Xiuxiu Lyu, M.Sc.
Dipl.-Ing. Jan Matheis
Daiki Matsumoto, M.Sc.
Lu Miao, M.Sc.
Matteo Moioli, M.Sc.
Dipl.-Phys. Christoph Niedermeier
Daria Ogloblina, M.Sc.
Raffaele Olmeda, M.Sc.
Aleksandra Pachalieva, M.Sc.
Ludger Pähler, M.Sc.
Shucheng Pan, M.Sc.
Thomas Paula, M.Sc.
Dr.-Ing. Albert Pernpeintner
Stefan Pfnür, M.Sc.
Dipl.-Ing. Julie Piquee
Patrick Pölzlbauer, M.Sc.
Dipl.-Ing. Jan-Frederik Quaatz
Vladyslav Rozov, M.Sc.
Dr.-Ing. Steffen Schmidt
Dipl.-Ing. Marco Stuhlpfarrer
Theresa Trummler, M.Sc.
Konstantin Vachnadze, M.Sc.
Jianhang Wang, M.Sc.
Zhaoguang Wang, M.Sc.
Josef Winter, M.Sc.
Dipl.-Ing. Maximilian Winter

Technical Staff
Martin Banzer
Franz Färber
Hans-Gerhard Frimberger
Wolfgang Lützenburg (Werkstattleiter)
Detlef Mänz
Hans-Jürgen Zirngibl

Research Foci
- Numerical fluid and flow modeling and simulation
- Complex fluids
- Turbulent and transitional flows
- Aerodynamics of aircraft and automobiles
- Environmental aerodynamics

Competences
- Multi-physics code and particle-based model development
- DrivAer car geometry
- Experimental aerodynamics

Infrastructure
- 3 low-speed wind tunnels and moving belt system
- 2 shock tubes

Courses
- Grundlagen der Fluidmechanik I
- Fluidmechanik II
- Computational Solid and Fluid Dynamics
- Aerodynamik des Flugzeugs I
- Aerodynamik des Flugzeugs II
- Grenzschichttheorie
- Angewandte CFD
- Gasdynamik
- Turbulente Strömungen
- Aerodynamik bodengebundener Fahrzeuge
- Aerodynamik der Bauwerke
- Aerodynamik von Hochleistungsfahrzeugen
- Instationäre Aerodynamik I
- Instationäre Aerodynamik II
- Numerische Berechnung turbulenter Strömungen
- Numerische Methoden für Erhaltungsgleichungen
- Aerodynamik der Raumfahrzeuge – Wiedereintrittsaerodynamik
- Particle-Simulation Methods for Fluid Dynamics
- Biofluid Mechanics
- Grundlagen der experimentellen Strömungsmechanik
- An Introduction to Microfluidic Simulations
- Physics of Fluids
- Strömungsphysik und Modellgesetze
- Praktikum Aerodynamik des Flugzeugs
- Praktikum Simulation turbulenter Strömungen auf HPC-Systemen
- Praktikum Experimentelle Strömungsmechanik

Aerodynamics and Fluid Mechanics

Selected Publications 2017

- Beban, B.; Schmidt, S.J.; Adams, N.A.: Numerical study of submerged cavitating throttle flows. Atomization and Sprays (27 (8)), 2017, pp. 723-739
- Buzica, Andrei; Bartasevicius, J.; Breitsamter, C.: Experimental investigation of high-incidence delta-wing flow control. Experiments in Fluids 58: 131, 2017
- Buzica, A.; Breitsamter, C.: Experimental and Numerical Investigation on delta-Wing Post-stall Flow Control. In: Diverse (Hrsg.): New Results in Numerical and Experimental Fluid Mechanics XI. Springer, 2017, pp. 167-179
- Diegelmann, F., Hickel, S., Adams, N.A.: Three-dimensional reacting shock–bubble interaction. Combustion and Flame 181, 2017, 1339-1351
- Fu, L.; Hu, X.Y.; Adams, N.A.: Single-step reinitialization and extending algorithms for level-set based multi-phase flow simulations. Computer Physics Communications (Article in press), 2017
- Fu, L.; Hu, X.Y.; Adams, N.A.: Targeted ENO schemes with tailored resolution property for hyperbolic conservation laws. Journal of Computational Physics (349), 2017, pp. 97-121
- Kaiser, J.; Adami, S.; Adams, N.A.: Direct Numerical Simulation of Shock-Induced Drop Breakup with a Sharp-Interface-Method. Symposium on Turbulence and Shear Flow Phenomena [TSFP10], 2017
- Kiewat, M.: Low-Memory Reduced-Order Modelling with Dynamic Mode Decomposition Applied on Unsteady Wheel Aerodynamics. American Society of Mechanical Engineers (ASME) Fluids Engineering Division Summer Meeting (FEDSM), 2017
- Kiewat, M.: Online Dynamic Mode Decomposition for the Investigation of Unsteady Vehicle Aerodynamics of the DrivAer Model: Part 2. Application on Velocity Fields. Annual Congress (Spring), 2017
- Knoth, F.; Breitsamter, C.: Flow Analysis of a Helicopter Engine Side Air Intake. Journal of Propulsion and Power, 2017
- Knoth, F.; Breitsamter, C.: Aerodynamic Characteristics of Helicopter Engine Side Air Intakes. Aircraft Engineering and Aerospace Technology, 2017
- Knoth, F.; Breitsamter, C.: Aerodynamic Testing of Helicopter Side Intake Retrofit Modifications. Aerospace (Vol. 4 (3)), 2017
- Pan, S.; Adami, S.; Hu, X.; Adams, N.A.: Shock-bubble Interaction Near a Compliant Tissue-like Material. International Symposium on Turbulence and Shear Flow Phenomena [TSFP 10], 2017
- Pasquariello, V.; Hickel, S.; Adams, N.A.: Unsteady effects of strong shock-wave/boundary-layer interaction at high Reynolds number. Journal of Fluid Mechanics (823), 2017, pp. 617-657
- Piquee, J.; Breitsamter, C.: Numerical and Experimental Investigations of an Elasto-Flexible Membrane Wing at a Reynolds Number of 280.000. Aerospace, 2017
- Piquee, J.; Saeedi, M.; Breitsamter, C.; Wüchner, R.; Bletzinger, K.-U.: Numerical Investigation of an Elasto-Flexible Membrane Airfoil Compared to Experiments. In: New Results in Numerical and Experimental Fluid Mechanics XI. Srpinger, 2017, pp. 421-433
- Rozov, V.; Hermanutz, A.; Breitsamter, C.; Hornung, M.: Aeroelastic Analysis of a Flutter Demonstrator with a very Flexible High-Aspect-Ratio Swept Wing. International Forum on Aeroelasticity and Structural Dynamics [IFASD], 2017
- Stemmer, C.; Birrer, M.; Adams, N.A.: Hypersonic boundary-layer flow with an obstacle in thermochemical equilibrium and nonequilibrium. Journal of Spacecraft and Rockets (54 (4)), 2017, pp. 899-915
- Stemmer, Christian; Birrer, Marcel; Adams, Nikolaus A.: Disturbance Development in an Obstacle Wake in a Reacting Hypersonic Boundary Layer. Journal of Spacecraft and Rockets (54(4)), 2017, pp. 945-960
- Winter, M.; Heckmeier, F.M.; Breitsamter, C.: CFD-Based Aeroelastic Reduced-Order Modeling Robust to Structural Parameter Variations. Aerospace Science and Technology (67), 2017, pp. 13-30
- Winter, M.; Breitsamter, C.: Application of Unsteady Aerodynamic Reduced-Order Modeling Techniques to a Complex Configuration. International Forum on Aeroelasticity and Structural Dynamics, 2017
- Winter, M.; Breitsamter, C.: Coupling of Recurrent and Static Neural Network Approaches for Improved Multi-step Ahead Time Series Prediction. In: Diverse (Hrsg.): New Results in Numerical and Experimental Fluid Mechanics XI. Springer, 2017, pp. 433-445
- Zhang, C.; Hu, X.Y.; Adams, N.A.: A weakly compressible SPH method based on a low-dissipation Riemann solver. Journal of Computational Physics 335, 2017, pp. 605-620
- Zhang, C.; Hu, X.Y.; Adams, N.A.: A generalized transport-velocity formulation for smoothed particle hydrodynamics. Journal of Computational Physics, 337, 2017, pp. 216-232

Flow Control and Aeroacoustics

Numerical and experimental study of flow and sound fields and their control

■ *The focus of the research group in 2017 was the development, testing and usage of research tools for the numerical prediction of flow and sound fields and the improvement of model-scale experiments in wind tunnels.*

Our research dealt with topics in three focus areas, including the numerical and experimental modeling of the wake evolution of wind turbines, the numerical and experimental modeling of self-noise from splitter attenuators and generic flow control studies. An achievement was the successful application for substantial computer resources for large-scale computations to be carried out with the LRZ facilities for high-performance computation on massively parallel computer systems. They form the basis for the ongoing investigations.

Control of a Three-dimensional Shear Layer by Oblique Vortices

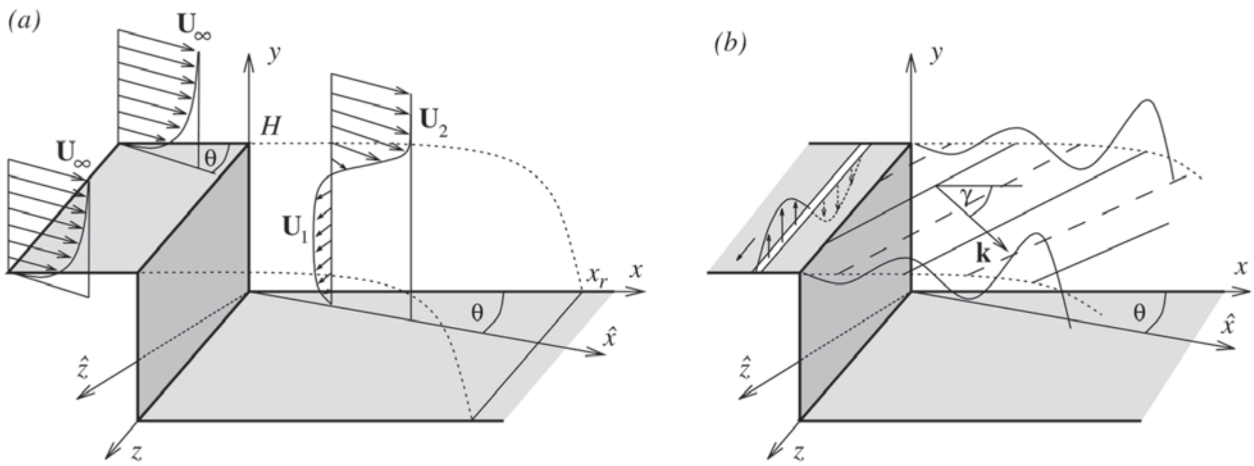


Figure 1: Generic geometry for the investigation of open-loop flow control by oscillatory blowing and suction along a strip aligned with the step edge.

An established open-loop flow control method for the mitigation of flow separation is oscillatory blowing and suction through holes or slots in the vicinity of the separation location on a body immersed in a flow. The artificial insertion of steady or propagating vortices enhances the momentum transfer across the shear layer. In order to better understand the role of forcing parameters numerical investigations have been carried out by means of large-eddy simulation for the generic configuration shown in Fig. 1. It consists of a turbulent boundary layer approaching a swept backward-facing step geometry. Vortices are generated by wall-normal blowing and suction along the step edge in terms of a wave propagating along the z-axis.

Using a suitable choice of the forcing parameters angular frequency and wave-length along the z-axis oblique vortices are generated that are primarily advected by the mean flow in the shear layer as shown in Fig. 2. Their mixing capability depends on their strength (related to the forcing amplitude), their spacing and their orientation. For the generic configuration a maximum efficiency in terms of shortening of the reattachment length was observed for a train of vortices with their axes deviating by an angle of 50° from the direction of the step edge. The achievement was similar for both the planar case with the mean flow normal to the step and for an inflow profile oriented in an angle of 40° towards the step.

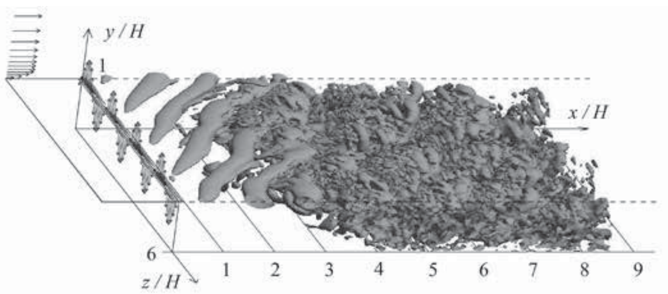


Figure 2: Isosurfaces of the second invariant of the velocity gradient tensor for turbulent flow across a backward-facing step subject to harmonic forcing by means of a wave propagating along the step edge.

Numerical and Experimental Modeling of the Influence of Ground Roughness on the Wake Evolution of Wind Turbines

In order to identify the power produced by a model-scale wind turbine (such as shown in Fig. 3) from the voltage and current of the attached generator, the mechanical and other losses of the drive train have to be known. Within the Eurotech collaboration on wind energy (Greentech project 03 Wind managed under the auspices of IGSSE) our partners at EPFL (group of Prof. F. Porte-Agel) offered us the possibility to calibrate the motors/generators for two model-scale wind turbines in the facilities of the WIRE lab at Lausanne which has recently acquired a suitable miniature torque meter. In combination with our force measurements these calibrations are mandatory for a proper determination of the dependence of thrust and power coefficients c_T and c_P as function of tip-speed ratio and yaw angle under different roughness conditions.

Figure 3: Model-scale wind turbine for the experimental investigation of the wake evolution over rough terrain in the atmospheric boundary layer wind tunnel of the Chair of Aerodynamics and Fluid Mechanics. The rotor diameter is abouit 450 mm.

Flow Control and Aeroacoustics

Prof. Dr.-Ing. Hans-Jakob Kaltenbach

Contact
www.aer.mw.tum.de
Hans-Jakob.Kaltenbach@tum.de
Phone +49.89.289.16397

Management
Prof. Dr.-Ing. Hans-Jakob Kaltenbach

Administrative Staff
Amely Schwörer
(secretary to the Chair of Aerodynamics)

Research Scientists
Dipl.-Ing. Victor Stein
Jithendra Tirakala, M.Sc.
Konstantin Vachnadze, M.Sc.

Research Focus
- Numerical prediction of generation and propagation of flow-induced noise
- Flow control with focus on suppression of flow separation and noise mitigation
- Self-noise of splitter attenuators
- Wake interaction of wind turbines

Competence
- Numerical prediction of flow and sound
- Experimental investigation of flow and sound fields

Infrastructure
- Usage of wind-tunnel at the Institute of Aerodynamics and Fluid Mechanics
- Test set-up of a microphone array

Courses
- Continuum Mechanics (for B.Sc. Engineering Sciences of MSE), 50%
- Grundlagen der numerischen Strömungsmechanik
- Aeroakustik
- Strömungsbeeinflussung
- Numerische Strömungsakustik
- Praktikum Numerische Strömungs-simulation
- Praktikum Numerische Strömungs-akustik

Publications 2017
- Jürgens, W. & Kaltenbach, H.-J.: Control of a three-dimensional shear layer by means of oblique vortices. Theor. Comput. Fluid Dyn. (2017). https://doi.org/10.1007/s00162-017-0447-8