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 2017-18 continued 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 new focus has been placed on learning effective evolution equations from data and physical knowledge, in cooperation with George Karniadakis who received the Humboldt research award in 2018.

A highlight in 2017-18 was the publication of a paper on bubble-collapse-driven penetration of biomaterial-surrogate liquid-liquid interfaces by Shucheng Pan, Stefan Adami, Xiangyu Hu and Nikolaus Adams, which was made the 'editor's choice' of Phys. Rev. Fluids. Nikolaus Adams was appointed as consultant professor and faculty member of the Northwestern Polytechnical University in Xi'an. Updates on the NANOSHOCK open-source code development are available for the scientific community: www.aer.mw.tum.de/abteilungen/nanoshock/news

Experimental Investigation of Shock-induced Droplet Break-up and Numerical Simulation of Collapsing Clouds of Vapor Bubbles

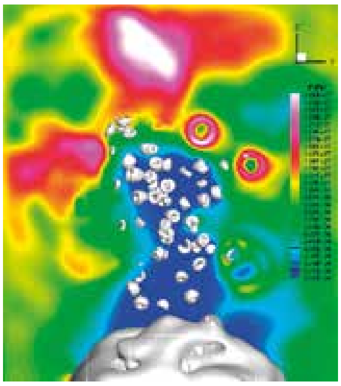
Motivation and Objectives

1) Shock-induced droplet break-up

The break-up of liquid droplets and fluid ligaments in a gaseous ambience is a key element of atomization processes. In combustion engines, the quality of the spray inside the combustion chamber has a large impact on the combustion efficiency and also on size and composition of particles in the exhaust gas. Furthermore, droplet break-up can play an important role in the production of metal powders as used for additive manufacturing. In this case, liquid metal atomization needs to be controlled in order to optimize the quality of the resulting powder. Our objective is to gain insight into break-up mechanisms by investigation of Newtonian and non-Newtonian liquid drops exposed to shock waves generated by a shock tube.

2) Collapsing clouds of vapor bubbles

It is well known that the collapse of vapor bubbles in a pressurized liquid can lead to intense pressure waves with amplitudes of several GPa. The formation of those bubbles can be on purpose, such as in biomedical applications and food engineering, or inevitable, such as in control valves of injection systems [1], rocket engines and in the vicinity of ship propellers. Since the release of potential energy during the collapse of a bubble can be highly focused, it may be used to destroy cancer cells. On the other hand, if clusters of bubbles collapse close to a material surface, severe damage of mechanical devices can be a consequence [2]. Our objectives are to develop and improve numerical techniques for prediction of vapor bubble collapses and to improve understanding of bubble-bubble interaction in collapsing vapor bubble clouds. Furthermore, experimental investigations are performed by exposing bubbles trapped in gelatin to shock waves generated by a shock tube [3-4].

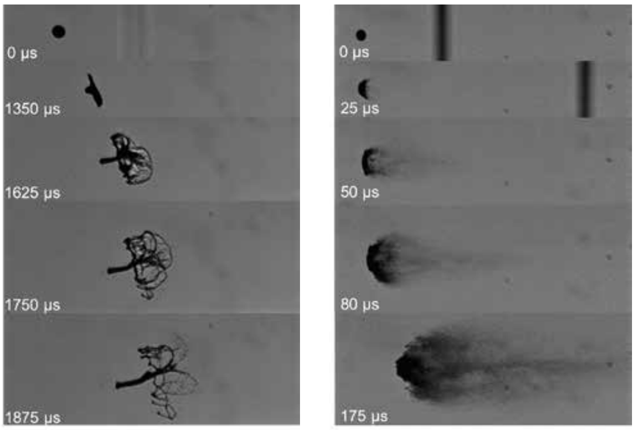


Collapsed and partially rebounded vapor bubbles and vapor pattern located at a solid surface (bottom). The colors indicate the shock-wave intensity due to prior collapse processes.

Approach to Solution

We develop and improve mathematical models and highly efficient numerical approaches for simulation of compressible multi-phase flows, especially physically consistent LES (large eddy simulation) codes. The codes are capable of high performance computations on supercomputers, such as SuperMUC at the Leibniz-Rechenzentrum München. The figure above shows collapsed and partially rebounded bubbles, together with a vapor pattern located at a solid surface. The colors indicate shock waves due to prior collapse processes. In this investigation [5], the effects of bubble interaction on intensification of material loads were characterized. It was possible to demonstrate that rebounding vapor patterns can be as erosive as the primary collapse of a bubble cloud.

The shock tube at the institute was recently equipped with a droplet generator in order to investigate shock-induced droplet break-up processes. State-of-the art high speed cameras/sensors allow for high-quality data acquisition. The following figure shows two time series of break-up processes. In both cases, the bubble is hit by a planar shock wave from left.



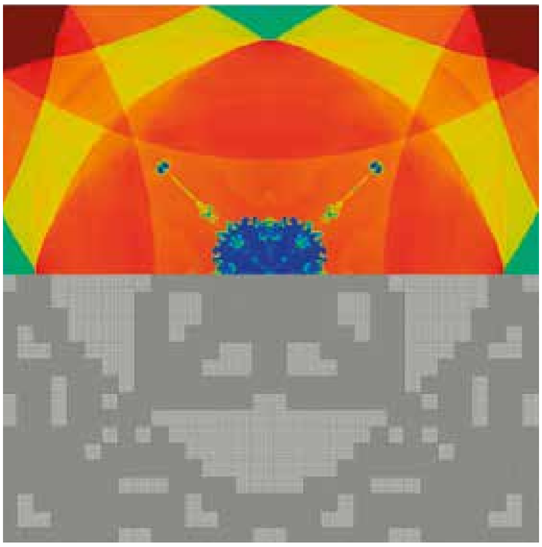
Shock-induced droplet break-up: bag-stamen type (left) and catastrophic type(right) for single water droplets (d=1mm) with Weber numbers of 33 and 1310, respectively.

Our research is funded by the European Union (project ‘CaFE’ and project ‘UCOM’), the European Space Agency, the German Research Foundation (DFG), and by partners from the automotive industry.

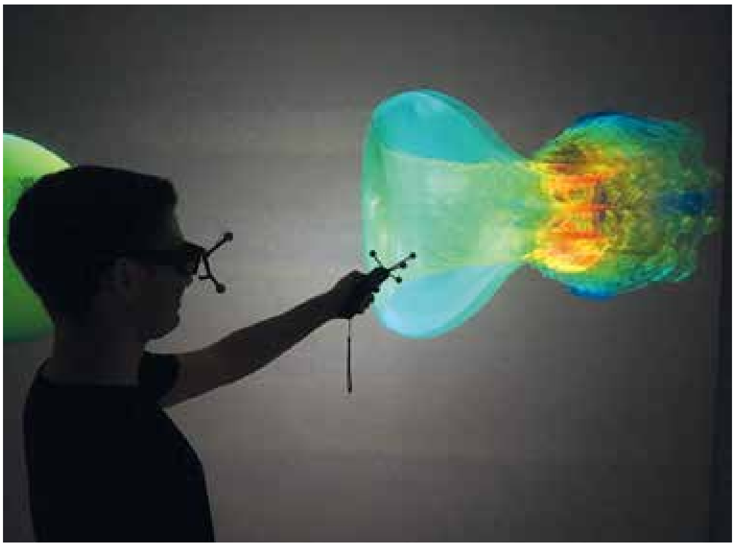
Key Results

1) Trummler, T.; Rahn, D.; Schmidt, S.J.; Adams, N.A.: LES of cavitating flow in a step nozzle with injection into gas. Atomization and Sprays, Vol. 28, Issue 10, 2018, pp 931-955
2) Gorkh, P.; Schmidt, S. J.; Adams, N. A.: Numerical investigation of cavitation-regimes in a converging-diverging nozzle. International Symposium on Cavitation, 2018
3) Wang, Z.; Hopfes, T.; Giglmaier, M.; Adams, N.A.: Influence of non-Newtonian gelatinous fluids on bubble collapse dynamics. International Conference on Experimental Fluid Mechanics, 2018
4) Hopfes, T.; Wang, Z.; Giglmaier, M.; Adams, N.: Experimental study on the effects of phase change during a bubble collapse. International Symposium on Cavitation, 2018
5) Ogloblina, D.; Schmidt, S. J.; Adams, N.A.: Numerical simulation of collapsing vapor bubble clusters close to a rigid wall. International Symposium on Cavitation, 2018

NANOSHOCK* – Manufacturing Shock Interactions for Innovative Nanoscale Processes



High-resolution simulation of an implosion problem: visualization of the pressure field and instantaneous grid representation showing the local refinement regions.



Shock-bubble interaction of an air-helium interface: 3D visualization of the late stage interface deformation at LRZ.

Motivation and Objectives

We want to investigate the potential of shockwaves for in-situ control of fluid processes with surgical precision. Shockwaves are discontinuities in the macroscopic fluid state that can lead to extreme temperatures, pressures and concentrations of energy. Applications of such shock interactions range from kidney-stone lithotripsy and drug delivery, to advanced aircraft design. With the use of properly focused shockwaves on tissue material, e.g. lesions with unprecedented surgical precision can be generated. Alternatively, improving combustion by enhanced mixing of fuels, shockwave interactions can help to further destabilize and atomize spray droplets.

** This project has received funding from the European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation programme (grant agreement No 667483).*

Aerodynamics and Fluid Mechanics

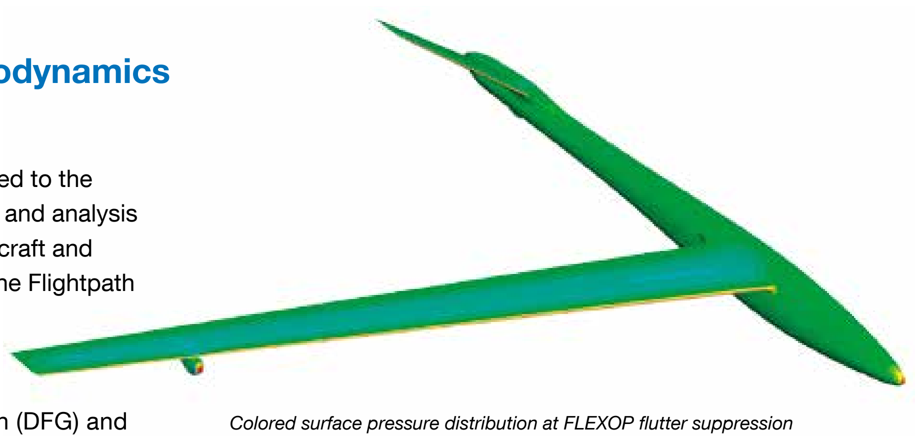
Our overall objective is to understand and predict the formation and control of shocks in complex environments, such as living organisms, using computational methods.

Approach to Solution
We develop best-in-class numerical methods with unprecedented accuracy and stability. The highly complex dynamics of shock-driven multiphase flows require very efficient numerical algorithms to handle the required mesh resolution. We have developed the simulation framework ALPACA that uses multiresolution techniques to compress the numerical grid without loss of accuracy. We use MPI-parallelization to perform efficient simulations on modern HPC architectures using large resolutions to capture all details at phase interfaces and flow discontinuities. With the new level-of-detail available in numerical simulations, we can better understand the underlying physics of complex multi-scale interactions. As a new feature, we now also support 3D visualization at the 5-sided projection installation at LRZ.
ALPACA is open-source and available to the public on request, see http://nanoshock.org or more information.

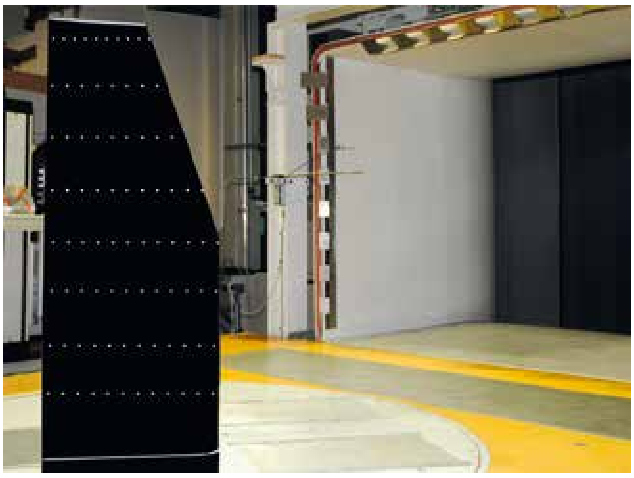
Key results
- Phenomenology of bubble-collapse-driven penetration of biomaterial-surrogate liquid-liquid interfaces, S. Pan, S. Adami, X. Hu, N.A. Adams, Physical Review Fluids, Vol. 3, Issue 2, 2018, Article Number 114005
- Droplet breakup as multi-scale computing challenge, S. Adami, N.A. Adams, invited talk at IUTAM Symposium on Dynamics and Stability of Fluid Interfaces, 2018
- Open-source version of ALPACA available to interested users

Aircraft and Helicopter Aerodynamics

Motivation and Objectives
The long-term research agenda is dedicated to the continued improvement of flow simulation and analysis capabilities enhancing the efficiency of aircraft and helicopter configurations with respect to the Flightpath 2050 objectives.
Aircraft aerodynamics research is aimed at detailing flow physics understanding of leading-edge vortex evolution (DFG) and vortex interaction effects (DFG) along with diamond wing aerodynamics and turbulence model conditioning (VitAM, LuFo V-3). Analysis of fluid-structure-interaction effects and aeroelasticity is linked to elasto-flexible lifting surface characteristics (DFG), flutter suppression techniques (FLEXOP, EU) and neuro-fuzzy based reduced order models addressing buffeting and buzz (DFG). Investigations on transport aircraft wings is dedicated to wake vortex alleviation and dynamic lift increase (BIMOD, LuFo V-3). The research work in the field of propeller and helicopter aerodynamics is related to propeller flow analysis at strong inhomogeneous inflow conditions (HyProp, BFS), propeller optimization strategies (AURAIS, Bay. LuFo) and full fairing rotor head design optimization of the RACER configuration (FURADO, CSky2). Further, 3D-printed advanced pressure probes including novel unsteady pressure sensors are under development (ZIM).



Colored surface pressure distribution at FLEXOP flutter suppression demonstrator configuration.



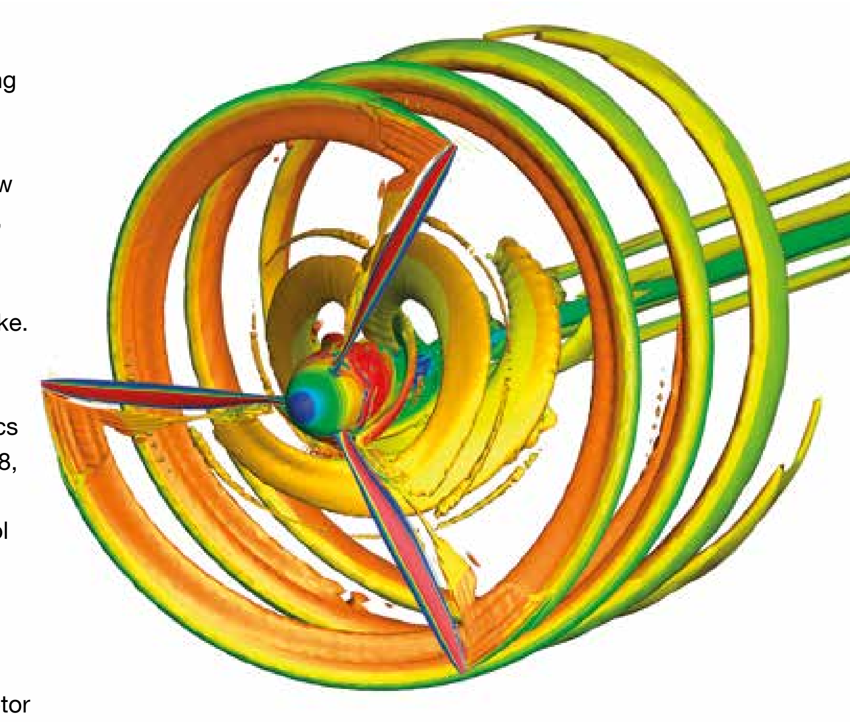
Test bed of an elasto-flexible membrane wing configuration mounted in the test section of wind tunnel A.

Approach to Solution
The investigations are conducted using both wind tunnel experiments and state-of-the art numerical simulations. In-house codes are continously further elaborated 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.

Aerodynamics and Fluid Mechanics

Key Results

- Buzica, A., Debschütz, L., Knoth, F., and Breitsamter, C.: Leading-edge Roughness Affecting Diamond Wing Aerodynamics, Aerospace, Vol. 5, 98, 2018, pp. 1-23
- Buzica, A. and Breitsamter, C.: Experimental and Numerical Investigation on Delta-Wing Post-stall Flow Control. NNFM, Vol. 136, Springer 978-3-319-64518, 2018, pp. 167-177
- Knoth, F. and Breitsamter, C.: Numerical and Experimental Investigation of a Helicopter Engine Side Intake. NNFM, Vol. 136, Springer 978-3-319-64518, 2018, pp. 27-39
- Pfnür, S., and Breitsamter, C.: Unsteady aerodynamics of a diamond wing configuration. CEAS Journal, 2018, DOI 10.1007/s13272-018-0280-9, pp. 1-20
- Pfnür, S., Oppelt, S., and Breitsamter, C.: Yaw-control efficiency analysis for a diamond wing configuration with outboard split flaps. CEAS Journal, 2018, DOI 10.1007/s13272-018-0340-1, pp. 1-19
- Pölzlbauer, P., Desvigne, D., and Breitsamter, C.: Aerodynamic Design Optimization of a Helicopter Rotor Blade-Sleeve Fairing. CEAS Journal, 2018
- 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 978-3-319-64518, 2018, pp. 421-431
- Winter, M., and Breitsamter, C.: Nonlinear identification via connected neural networks for unsteady aerodynamic analysis. Aerospace Science and Technology, Vol. 77, 2018, pp. 802-818



Computed (left) and measured (Stereo-PIV measurements for inhomogeneous inflow, right) propeller flow field.

- 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, 2018, pp. 433-442

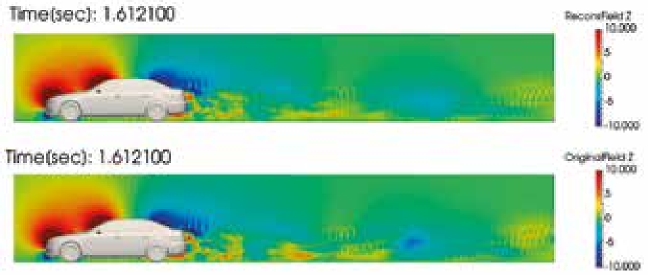
Reduced Order Modeling for Automotive Aerodynamics

The recent improvement of high-performance computing hardware has enabled the utilization of unsteady computational fluid dynamics (CFD) for industrial product development. Unsteady CFD can accurately simulate the transient phenomena of the flow field, moreover, highly accurate steady-state results can also be obtained through appropriate averaging. Especially in the field of automotive aerodynamics, the transient flow phenomena around the vehicle can strongly affect driving stability and ride comfort. A difficulty in the analysis of the transient flow field by CFD is that the time series of flow field data typically needs to be saved to disk during and after a simulation. This often requires massive storage space, as the transient flow field data around a vehicle is spatially highly resolved to capture complex flow structures consisting of various time and length scales. One possible solution to reduce the total amount of data is to approximate a transient flow field

40% scale DrivAer model (notchback configuration) in wind channel A

in reduced order. Proper orthogonal decomposition (POD) is a well-known data-driven modal analysis method that is often used for reduced-order modeling of the flow field. Especially the on-the-fly algorithm of POD, such as incremental proper orthogonal decomposition

(IPOD) or incremental singular value decomposition, requires much less RAM and disk space, since it updates modes incrementally when new snapshot data is available.
As an example, the IPOD modes are computed from the simulated transient flow field around the DrivAer notchback model. The numerical simulation is validated with the wind tunnel experiment (figure above). The computed IPOD modes successfully approximate the complex transient flow field around the vehicle with respect to both transient characteristics and instantaneous flow structures (figure below). Furthermore, this unsteady flow field data approximated in reduced order can be processed by dynamic mode decomposition (DMD) to extract the dominant transient flow structures.
Consequently, the amount of transient flow field data is reduced to 11% of the original size with the setups presented. This IPOD computation occupies roughly 80% less memory than the conventional POD algorithm.



A snapshot of the instantaneous vertical velocity on the x-z-plane at y=0.2m from the reconstructed field (above) and the original field (below).

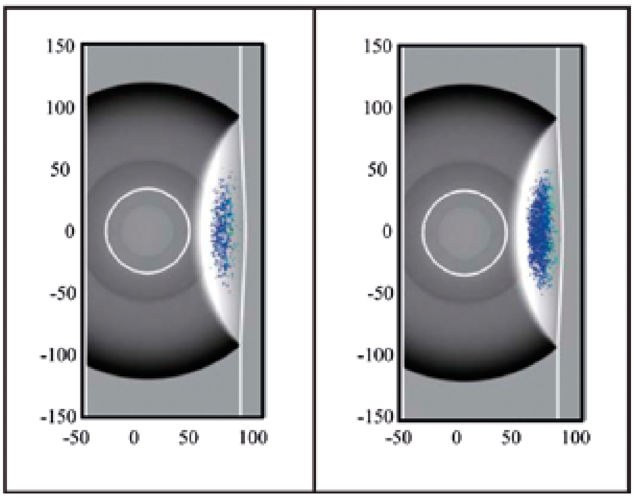
Publications

- Matsumoto, D., Kiewat, M., Niedermeier, C., & Indinger, T. (2018). Reduction of transient flow field data using Incremental Proper Orthogonal Decomposition. Proceedings in 2018 JSAE Annual Congress (Spring)
- Matsumoto, D., Kiewat, M., Haag, L., & Indinger, T. (2018). Online Dynamic Mode Decomposition Methods for the Investigation of Unsteady Aerodynamics of the DrivAer Model (First Report). International Journal of Automotive Engineering, Vol. 9, Issue 2, pp. 64-71
- Kiewat, M., Matsumoto, D., Haag, L., Zander, V., & Indinger, T. (2018). Online Dynamic Mode Decomposition Methods for the Investigation of Unsteady Aerodynamics of the DrivAer Model (Second Report). International Journal of Automotive Engineering, Vol. 9, Issue 2, pp. 72-78
- Haag, L., Kiewat, M., Indinger, T., & Blacha, T. (2017, July). Numerical and experimental investigations of rotating wheel aerodynamics on the DrivAer model with engine bay flow. In ASME 2017 Fluids Engineering Division Summer Meeting (pp. V01BT12A005-V01BT12A005). American Society of Mechanical Engineers
- Collin, C., Mueller, J., Islam, M., & Indinger, T. (2017). On the Influence of Underhood Flow on External Aerodynamics of the DrivAer Model. Proceedings of the 11th FKFS Conference, Progress in Vehicle Aerodynamics and Thermal Management, Stuttgart, Germany, pp. 201-215, ISBN 978-3-319-67822-1

Numerical Investigation of Homogeneous Cavitation Nucleation in a Microchannel

Motivation and Objectives

Liquid and gas (or vapor) two-phase flows are widely encountered in many chemical, biological, and engineering applications; here bubble cloud dynamics are of fundamental importance. Examples reach from ultrasonic cleaning through medical therapy applications to bacteria disinfection processes. In such flows, bubble nucleation, initializing liquid-to-vapor transition, can be categorized into heterogeneous and homogeneous nucleation. These differ with respect to where nucleation occurs. The former emerges from surfaces in contact with two liquids, the latter relies on impurities in the bulk liquid and thus is more difficult to localize and detect in experiments.



Numerical simulation of homogeneous nucleation in comparison with the experimental results.

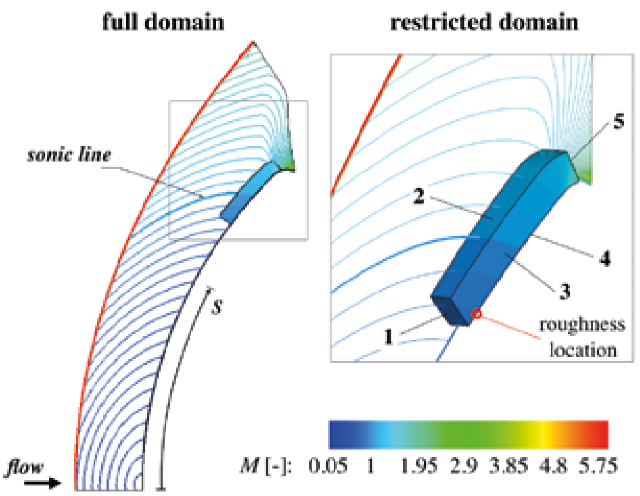
Approach to Solution

We investigate numerically homogeneous nucleation in a microchannel induced by shock reflection to gain a better understanding of the mechanism of homogeneous nucleation. The liquid expands due to the reflected shock and homogeneous cavitation nuclei are generated. An Eulerian-Lagrangian approach is employed for modeling this process in a micro-channel.

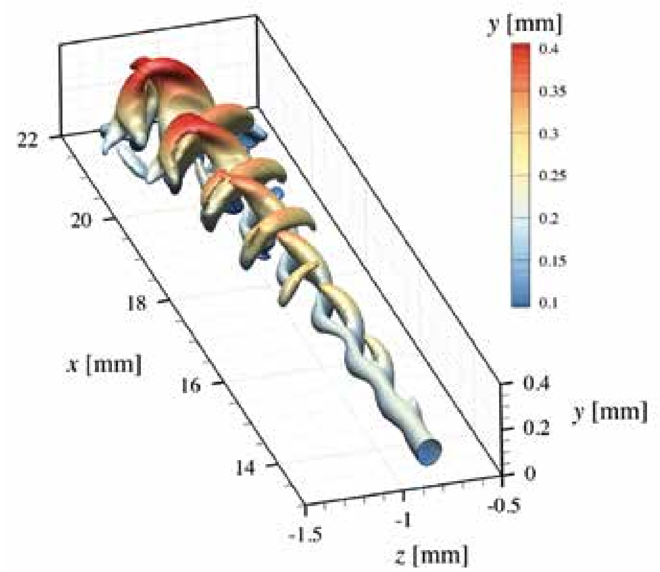
Key Results

- A conservative interface-interaction method for compressible multi-material flows. S. C. Pan, X.Y. Hu, N. A. Adams, Journal of Computational Physics, Vol. 371 (2018), pp. 870-895
- Numerical investigation of homogeneous cavitation nucleation in a microchannel. X.X. Lyu, S. C. Pan, X.Y. Hu, N. A. Adams, Physical Review Fluids, Vol. 3, Issue 6, (2018) 064303
- A Consistent Analytical Formulation for Volume Estimation of Geometries Enclosed by Implicitly Defined Surfaces. S. C. Pan, X.Y. Hu, N. A. Adams, SIAM Journal on Scientific Computing, Vol. 40, Issue 3 (2018) A1523-A1543

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



Integration domain on a re-entry capsule



Vortical structures and breakdown downstream of a roughness patch for re-entry condition

Motivation

Blunt bodies returning from space are subject to immense heat loads leading to ablation. Roughness on these ablating surfaces can induce laminar-turbulent transition in an otherwise laminar flow. Laminar-turbulent transition increases the heat load on the surface. Roughness is an enhancing effect on laminar-turbulent transition and the effect of roughness including dissociation and non-equilibrium effects is the focal point of the studies.

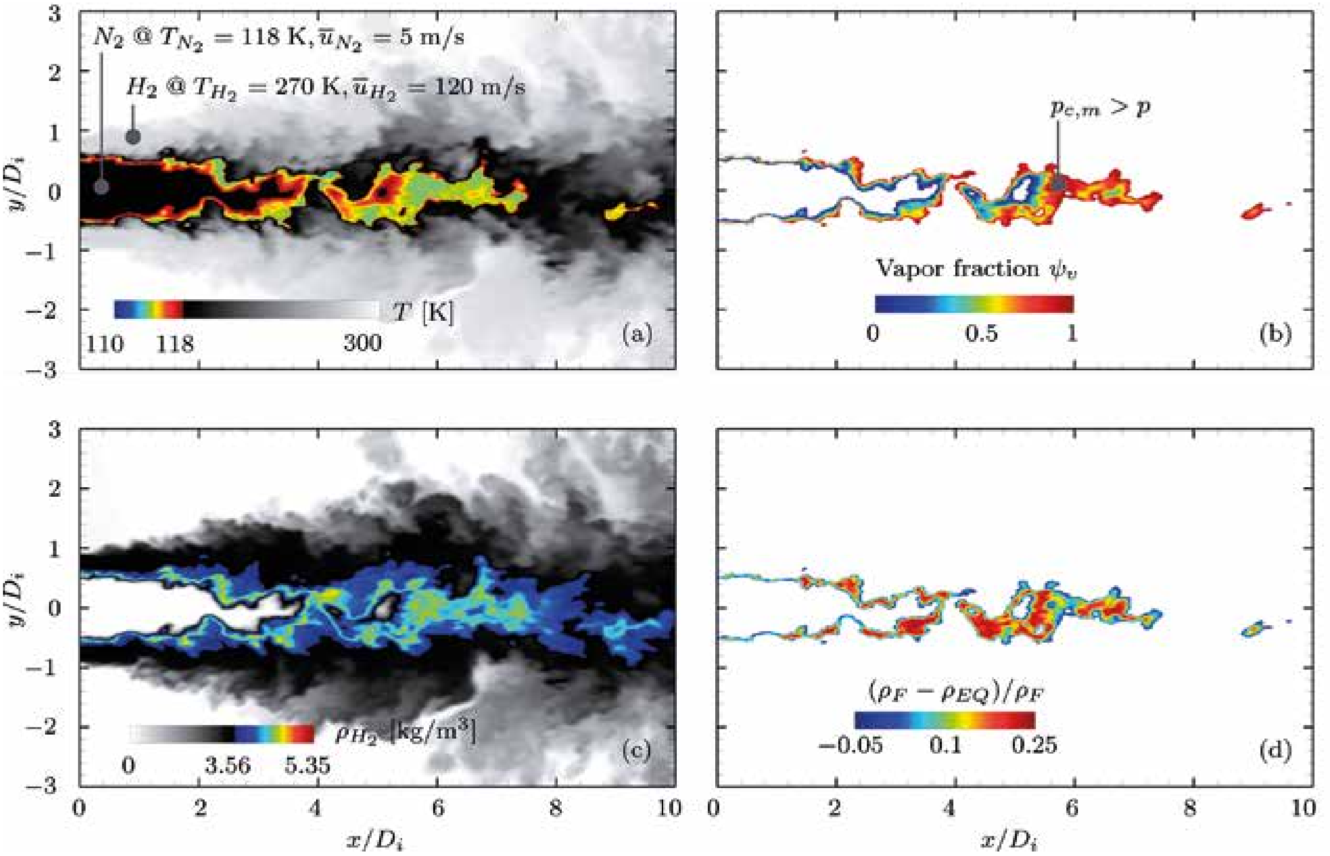
Approach to Solution

Direct numerical simulations (DNS) including hundreds of millions of points are conducted on national HPC facilities such as SuperMUC and HLRS. The results are compared in international cooperation with theoretical and experimental results from universities and research establishments worldwide.

Key Results

- A. Di Giovanni and C. Stemmer: Crossflow-type vortex breakdown in a hypersonic re-entry capsule boundary-layer. Journal of Fluid Mechanics, Vol. 856, 2018, pp. 470-503 http://dx.doi.org/doi.org/10.1017/jfm.2018.706
- S. Hein, A. Theiss, A. Di Giovanni, C. Stemmer, T. Schilden, W. Schröder, P. Paredes, M.M. Choudhari, F. Li, and E. Reshotko: Numerical Investigation of Roughness Effects on Transition on Spherical Capsules. Journal of Spacecraft and Rockets, 2019

DFG Sonderforschungsbereich TRR 40: Technological Foundations for the Design of Thermally and Mechanically Highly Loaded Components of Future Space Transportation Systems



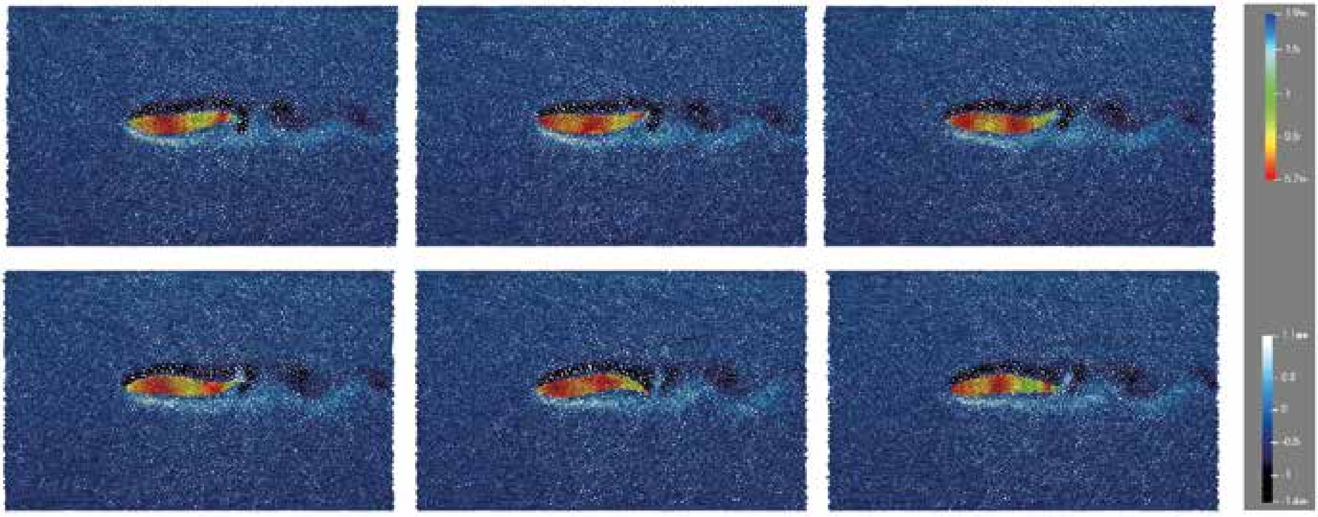
Instantaneous snapshots of a nitrogen jet in hydrogen: (a) temperature, (b) vapor fraction on a molar basis, (c) hydrogen density and (d) relative difference in density.

The Institute of Aerodynamics and Fluidmechanics has the speaker role within the DFG-SFB TRR40. Next-generation space transportation systems will be based on rocket propulsion systems which deliver the best compromise between development and production cost and performance. The TRR40 focuses on liquid rocket propulsion systems and their integration into the space transportation system.

Critical, thermally and mechanically highly loaded components of such space transportation systems are the combustion chamber, the nozzle, the aft body and the cooling of the structure. These components offer the highest potential for the efficiency increase of the entire system. However, all components are in close and direct interaction with each other. Optimization or the fundamentally new design of a single component directly affects all other components.

The 25 projects from TUM, RWTH Aachen, TU Braunschweig and the University of Stuttgart, as well as partners from DLR and AIRBUS D&S, investigate in interdisciplinary experimental and numerical teams. The concepts developed will be tested on sub-scale combustion chambers and will be developed to a stage of applicability. In addition, principal experiments are going to be conducted to demonstrate new technologies developed in the TRR40. The scientific focus of all five research areas within the TRR40 is the analysis and the modeling of coupled systems. Based on reference experiments detailed numerical models are developed which serve as the basis for efficient and reliable predictive simulation tools for design.

SPH Modeling of Fluid-Structure Interaction



The sequence of results (ordered left to right and top to bottom) shows an SPH simulation of a tethered fish flapping in a stream with a Reynolds number of 1,000. An inextensible rope is connected to the left boundary. The color presents velocity magnitude of the fish body and vorticity in flow.

Motivation and Objectives

Fluid-structure interaction (FSI) can be found in many natural phenomena, such as birds flying and fish swimming. Meanwhile, it also plays a very important role in a wide range of engineering areas, e.g. aeronautical engineering, coastal engineering and biomedical engineering. The essential of FSI is the interaction between movable or deformable structures with internal or surrounding fluid flows.

Approach to Solution

We propose a numerical modeling of FSI (fluid-structure interaction) problems in a unified SPH (smoothed particle hydrodynamics) framework. Rather than being strictly monolithic, the present modeling is the combination of a conventional SPH formulation for fluid motions and a total Lagrangian SPH formulation dealing with the structure dynamics. Since both fluid and solid governing equations are still solved with SPH algorithms, fluid-structure coupling is straightforward and the momentum conservation of an FSI system is strictly satisfied. Furthermore, the application of a Lagrangian kernel eliminates the particle-distribution artifact which exhibits in previous SPH simulation of structure dynamics using the incremental constitutive model. Several tests including pure structure oscillation and FSI benchmark cases have been carried out to validate the present modeling and demonstrate its potential.

Key Results

- SPH modeling of fluid-structure interaction, C. Zhang, X. Y. Hu. Journal of Hydrodynamics, Vol. 30 (2018), pp. 62-69
- Liquid Splash Modeling with Neural Networks. K. Um, X. Y. Hu, N. Thuerey. ACM SIGGRAPH/Eurographics Symposium on Computer Animation 2018, Computer Graphics Forum, Vol. 37, Issue 8
- An incremental-stencil WENO reconstruction for simulation of compressible two-phase flows. B. Wang, G. Xiang, X.Y. Hu. International Journal of Multiphase Flow, Vol. 104 (2018), pp. 20-31

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
PD Dr.-Ing. Habil. 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

Administrative Staff
Angela Grygier
Amely Schwörer
Dipl.-Des. Sabine Kutscherauer

Guest
Dr.-Ing. Rongzong Huang
Dr.-Ing. Yuxuan Zhang

Research Staff
Dr.-Ing. Stefan Adami
Vladimir Bogdanov, M.Sc.
Morgane Borreguero, M.Sc.
Aaron Buhendwa, M.Sc.
Dipl.-Ing. Andrei Buzica
Michael Cerny, M.Sc.
Antonio Di Giovanni, M.Sc.
Alexander Döhring, M.Sc.
Nico Fleischmann, M.Sc.
Dr.-Ing. Marcus Giglmaier
Polina Gorkh, M.Sc.
Florian Heckmeier, M.Sc.
Thomas Hopfes, M.Sc.
Nils Hoppe, M.Sc.
Naeimeh Hosseini, M.Sc.
Marian Izsak, M.Sc.
Zhe Ji, M.Sc.
Jakob Kaiser, M.Sc.
Dipl.-Ing. Thomas Kaller
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.
Daiki Matsumoto, M.Sc.
Lu Miao, M.Sc.
Matteo Moioli, M.Sc.
Dipl.-Phys. Christoph Niedermeier
Raffaele Olmeda, M.Sc.
Aleksandra Pachalieva, M.Sc.
Ludger Pähler, M.Sc.
Thomas Paula, M.Sc.
Dr.-Ing. Albert Pernpeintner
Jonathan Pflüger, M.Sc.
Stefan Pfnür , M.Sc.
Dipl.-Ing. Julie Piquee
Patrick Pölzlbauer, M.Sc.
Jan Reiß, M.Sc.
Vladyslav Rosov, M.Sc.
Johannes Ruhland, M.Sc.
Dr.-Ing. Steffen Schmidt
Julius Stegmüller, M.Sc.
Dipl.-Ing. Marco Stuhlpfarrer
Theresa Trummler, M.Sc.
Jianhang Wang, M.Sc.
Zhaoguang Wang, M.Sc.
Josef Winter, M.Sc.
Dipl.-Ing. Maximilian Winter
Chi Zhang, M.Sc.
Yujie Zhu, M.Sc.
Christopher Zöller, M.Sc.

Technical Staff
Martin Banzer
Luigi Findanno
Hans-Gerhard Frimberger
Wolfgang Lützenburg (Workshop manager)
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 Hochleistungs-fahrzeugen
- Instationäre Aerodynamik I
- Physik der Fluide
- Numerische Methoden für Erhaltungs-gleichungen
- Aerodynamik der Raumfahrzeuge – Wiedereintrittsaerodynamik
- Particle-Simulation Methods for Fluid Dynamics
- Biofluid Mechanics
- Grundlagen der experimentellen Strömungsmechanik
- An Introduction to Microfluidic Simulations
- Instationäre Aerodynamik II
- Strömungsphysik und Modellgesetze
- Praktikum Aerodynamik des Flugzeugs
- Praktikum Simulation turbulenter Strömungen auf HPC-Systemen
- Praktikum Experimentelle Strömungs-mechanik

Flow Control and Aeroacoustics

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

■ *The focus of the research group in 2018 was the development, testing and usage of research tools for the numerical prediction of flow and sound fields.*

Our research dealt with topics in two focus areas, including the numerical and experimental modeling of the wake evolution and radiation of low-frequency sound from wind turbines and tonal noise prediction for a 2-blade pusher propeller. For the wind-turbine flow simulation the in-house code INCA is used with actuator line treatment of rotating blades and LES modeling of the turbulent inflow and wake development over rough ground. For sound prediction the acoustic prediction tool SPYSI, developed at the Friedrich-Alexander Universität Erlangen in the group of Prof. S. Becker, is used. It is a time-domain implementation of the Ffowcs-Williams Hawkings formulation of Lighthills acoustic analogy. For the present studies it has been complemented in order to consider the effect of mean flow advection on the sound propagation. It has also been used to assess the low-frequency noise from wind turbines emerging from the unsteady blade loading due to interaction with atmospheric boundary layer turbulence.

Implementation of the Actuator Line Method

For upcoming studies on the evolution of wind turbine wakes in complex terrain the actuator line method was implemented in ANSYS Fluent. The implementation was compared with results from established codes like EllipSys-3D and INCA. Figure 1 shows the influence of the chosen turbulence model on the normal and tangential forces along the radial extent of the blade. Figure 2 visualizes the axial velocity field. The decay of the tip vortices turns out to be very sensitive with respect to details of the numerical solution scheme and chosen turbulence model.

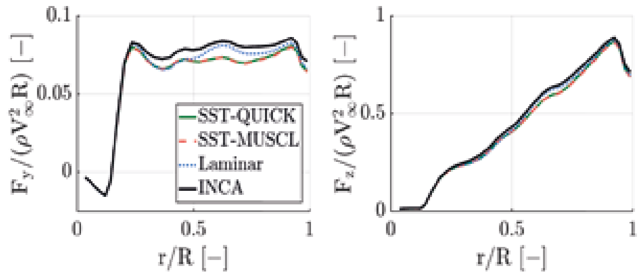


Figure 1: Tangential (left) and normal (right) component of the blade force along the radial extension of the blade for different turbulence models. From: Andrea Martinez-Garcia, term paper at SBA, TUM, 2018

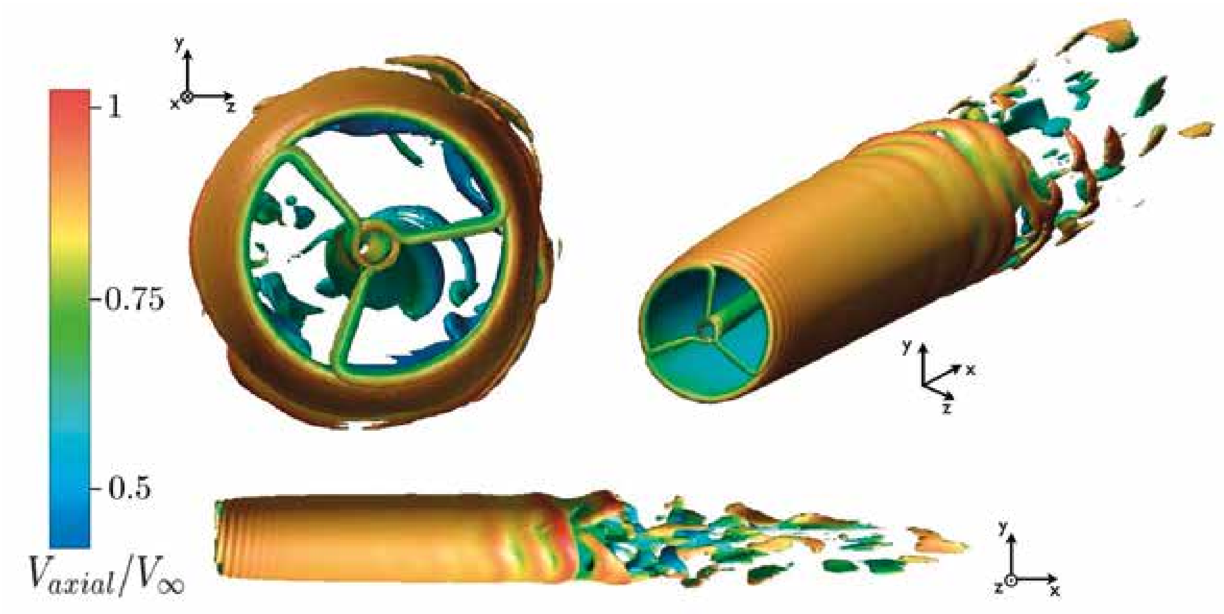


Figure 2: Isosurfaces of the axial component of the velocity in a wind turbine wake modelled in FLUENT with the ALM method. From: A. Martinez-Garcia, term paper, 2018

Influence of a V-shaped Tail on the Noise Radiation from a Two-bladed Pusher Propeller

The blade loading for a pusher propeller mounted downstream of the V-tail of the UAV IMPULLS has been predicted using ANSYS CFX in URANS mode. Figure 3 shows a side-view of the tail section. The propeller cuts through the wakes of the two stabilizer fins. As a result of the unsteady blade loading the contributions from higher harmonics of the blade passing frequency to the loading noise increases as shown in Figure 3 for an observer located 45 degrees off the rotor axis.

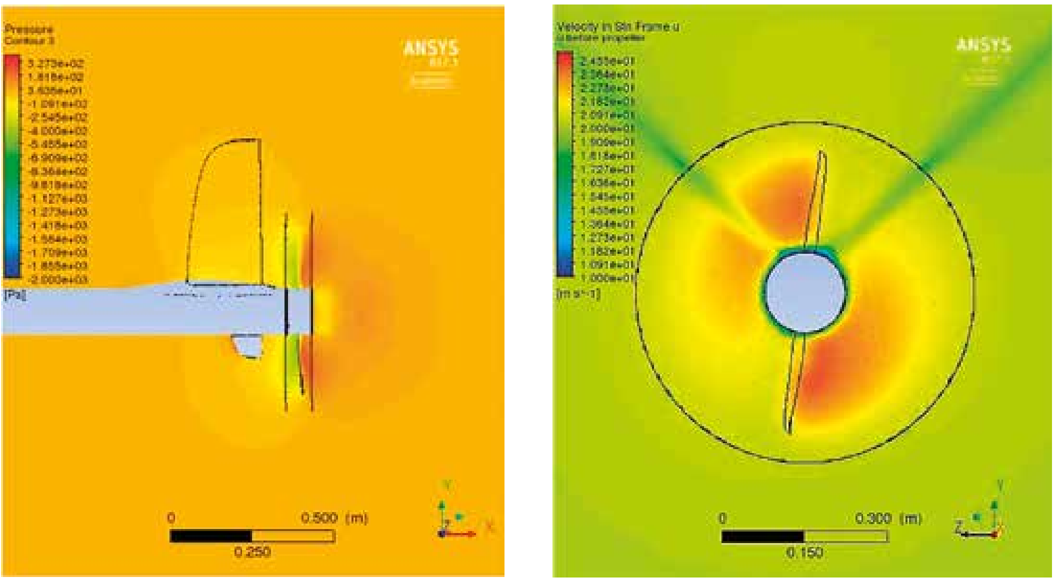


Figure 3: Left: Side-view of instantaneous static pressure in the tail section of the IMPULLS UAV equipped with a V-tail (empennage) and a two-bladed pusher propeller. Right: Magnitude of velocity in a cross-section between tail fins and propeller. Source: Peng Liu, Master's thesis, SBA, TUM 2018

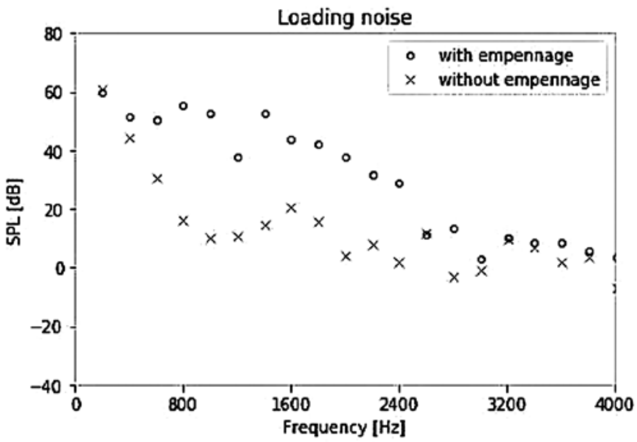


Figure 4: Comparison of loading noise in the far field at an observer position 45 degree off the axis from the pusher propeller with and without the empennage. Source: Peng Liu, Master's thesis, SBA, TUM 2018.

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
Marian Izsak, M.Sc.
Julius Stegmüller, M.Sc.
Dipl.-Ing. Victor Stein
Jithendra Tirakala, 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
- Use of wind tunnel at the Institute of Aerodynamics and Fluid Mechanics
- Test set-up of a microphone array

Courses
- Continuum Mechanics (for BSc Engineering Sciences of MSE), 50%
- Grundlagen der numerischen Strömungsmechanik
- Aeroakustik
- Strömungsbeeinflussung
- Numerische Strömungsakustik
- Praktikum Numerische Strömungssimulation
- Praktikum Numerische Strömungsakustik

Publications 2018

- Jürgens, W., Kaltenbach, H.-J.: Control of a three-dimensional shear layer by means of oblique vortices. Theor. Comput. Fluid Dyn., vol. 32 (2), 2018, pp. 179-199.
- Stein, V.P., Kaltenbach, H-J.: Influence of ground roughness on the wake of a yawed wind turbine – a comparison of wind-tunnel measurements and model predictions. Journal of Physics, Conference Series, vol. 1037 (7), 2018, article 072005.