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-surrrogate 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

### 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].

### 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.
Aerodynamics and Fluid Mechanics

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.

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.

Key Results

NANOSHOCK* – Manufacturing Shock Interactions for Innovative Nanoscale Processes

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 667463).
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**
- 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).

**Approach to Solution**
The investigations are conducted using both wind tunnel experiments and state-of-the art numerical simulations. In-house codes are continuously 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.
Key Results


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 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**


**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.**
Aerodynamics and Fluid Mechanics

**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**

---

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

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.
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**

Aerodynamics and Fluid Mechanics

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

Management
Prof. Dr.-Ing. Nikolaus A. Adams, Director
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 Gigmaier
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 Permeintner
Jonathan Pflüger, M.Sc.
Stefan Pfnür, M.Sc.
Dipl.-Ing. Julie Pique
Patrick Polzlhuber, 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 März
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 von Hochleistungs- fahrzeugen
- Instationäre Aerodynamik I
- Physik der Fluide
- 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
- Instationäre Aerodynamik II
- Strömungspfysik und Modellgesetze
- Praktikum Aerodynamik des Flugzeugs
- Praktikum Simulation turbulenter Strömungen auf HPC-Systemen
- Praktikum Experimentelle Strömungsmechanik
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 Flowcs-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.
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.
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 Strömungskonstruktion
- Praktikum Numerische Strömungssimulation
- Praktikum Numerische Strömungskonstruktion

Publications 2018