Aerodynamics and Fluid Mechanics

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

Prof. Dr.-Ing.
Nikolaus A. Adams

Contact

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

The focus of the Institute of Aerodynamics and Fluid Mechanics in 2015-16 was on propulsion fluid dynamics, deriving new particle methods for continuum mechanics, development of a multi-resolution parallel simulation environment for the NANOSHOCK project, on reduced-order modeling and control of rotary and fixed-wing aerodynamics, and on advanced simulation technologies in automotive aerodynamics.

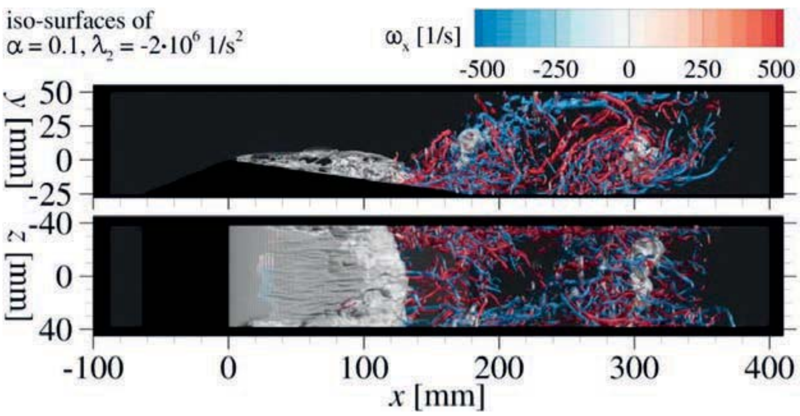
A highlight in 2015/16 was the successful defense of the Collaborative Research Program TRR 40 on propulsion systems for space transportation which entered its 3rd and final funding period 2016-2020. The TRR 40 is one of the largest fundamental research activities on this subject worldwide. Dr. Daniel Gaudlitz, formerly Senior Research Associate with the Institute, was appointed as Professor of Fluid Mechanics at the University of Applied Sciences in Zwickau.

Cavitation and Flow-Induced Erosion

Motivation and Objectives

Flow-induced evaporation (cavitation) of liquids occurs in a broad range of technical systems. In combustion engines, cavitation is leveraged in order to control the mass flow and to clean spray holes from exhaust products. Collapsing vapor bubbles may also be used to enhance drug delivery in biomedical applications. However, if vapor cavities collapse in an uncontrolled way, the surrounding material can be severely damaged. Especially for high pressure systems, such as fuel injector components of automotive and naval combustion engines, cavitation erosion represents a continuous challenge for designers. Violent collapses of vapor patterns can result in the formation of intense shock waves with amplitudes reaching more than 10,000 bar – enough to damage even stainless steel. In addition to material removal, cavitation may also induce strong noise and vibrations. Both are highly undesirable for pumps and turbines, as well as for naval vessels such as cruise liners.

Our objective is to implement accurate simulation approaches for predicting all dominating phenomena in cavitating flows, with the goal to provide the groundwork for the design optimization of future technical devices.



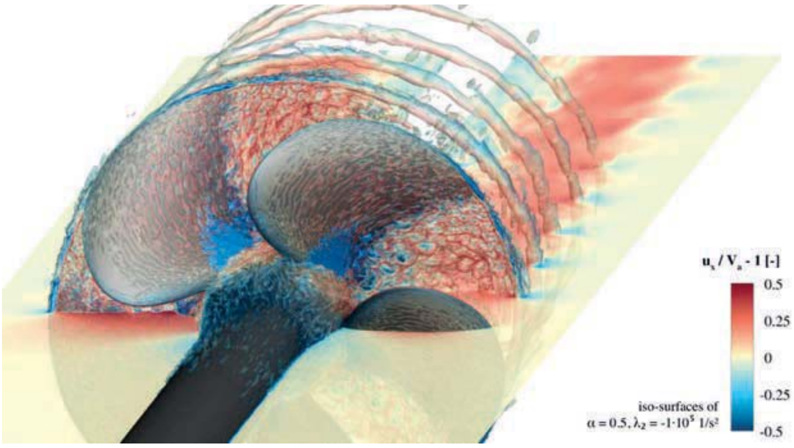
Side- and top-view on a shedding partial cavity forming past a sharp wedge.

Approach to Solution

We develop mathematical models and numerical approaches for efficient and accurate prediction of cavitating flow phenomena. The thermodynamic description of all fluid components (liquid, vapor, inert gases) involved makes for compressible fluid models possible. These enable the simulation of shock wave formation due to collapsing vapor patterns. Thereby, intense loads on material surfaces are obtained without the need for additional models. Depending on the dominant physics, high-quality shock-capturing schemes and large-eddy simulation (LES) schemes are proposed and applied to fundamental as well as to highly practical problems. Fundamental research is funded by the European Union, while applied research is performed in collaboration with automotive suppliers, the U.S. Office of Naval Research and the European Space Agency.

Key Results

- 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.
- Budich, B.; Schmidt, S.J.; Adams, N.A.: Implicit large-eddy simulation of the cavitating model propeller VP1304 using a compressible homogeneous mixture model. Symposium on Naval Hydrodynamics [31st], 2016.



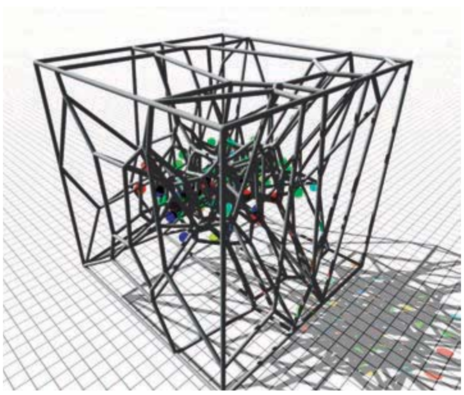
Cavitating ship propeller flow around the model propeller VP1304.

- Budich, B.; Schmidt, S.J.; Adams, N.A.: Numerical investigation of condensation shocks in cavitating flow. Symposium on Naval Hydrodynamics [31st.], 2016.

Particle Hydrodynamics for Domain Partition

Motivation and Objectives

Massively parallel computing is essential for computational fluid dynamics (CFD) to cope with flow simulations involving complex geometries or a wide spectrum of length scales. However, partitioning problems arise when the number of processor cores increases in massively parallel computing environments. The optimization strategy for load-balancing and inter-processor communication becomes the critical bottleneck and limits the computational performance.



The partitioning topology of partitioning number 64 for a 3D partitioning simulation.

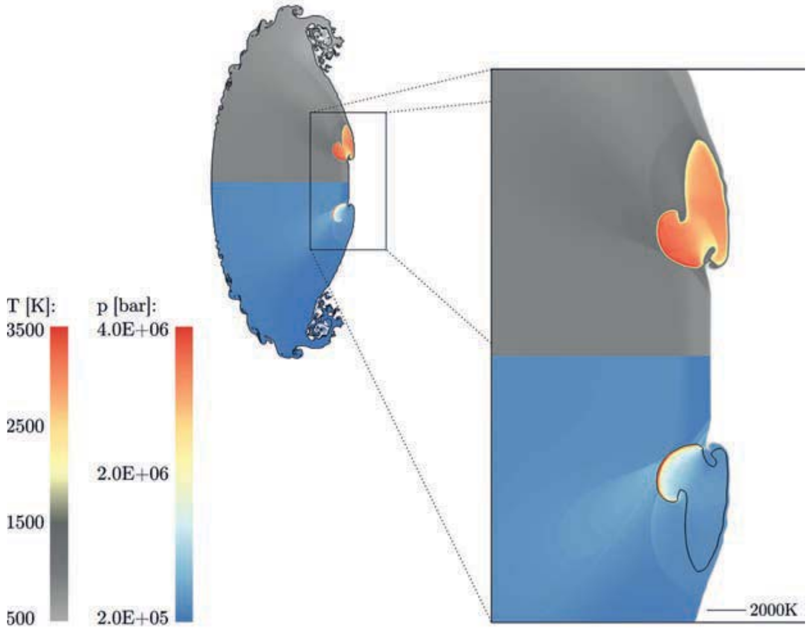
Approach to Solution

We have developed particle-hydrodynamics-based domain partition methods. Such methods are able to accomplish approximately equal-sized domain decomposition with minimum neighbor communication patterns as it reduces processor operation idle time and inter-processor communication time.

Key Results

- A novel partitioning method for block-structured adaptive meshes, Part I: Theory and validation, L. Fu, S. Litvinov, X.Y. Hu, N.A. Adams. Journal of Computational Physics, accepted for publication 2016.
- A physics-motivated centroidal voronoi particle domain decomposition method. L. Fu, X.Y. Hu, N.A. Adams. Journal of Computational Physics, accepted for publication 2016.
- A weakly compressible SPH method based on a low-dissipation Riemann solver. C. Zhang, X.Y. Hu, N.A. Adams. Journal of Computational Physics, accepted for publication 2016.

Large Eddy Simulation of Complex Turbulent Flows



Deflagration-to-detonation transition in a shock-induced shock-bubble interaction

Motivation and Objectives

Scientific discovery through modeling and predictive simulation requires numerical models and solution methods that accurately represent and resolve relevant flow physics and efficiently exploit modern massively parallel supercomputers.

Approach to Solution

Adaptive flow simulation refers to highly automated flow simulations which require only a minimum of user interventions, while delivering reliable results. This includes methods for automatic mesh adaptation as well as for the quantification of uncertainties that result from model approximations, initial data, or boundary conditions. Adaptive DNS/LES requires methods that couple numerical and physical models on multiple scales in a consistent way. We have developed sophisticated wall turbulence models, which facilitate LES of engineering aerodynamic applications.

Advanced coupling of fluid dynamics with complex chemical reaction kinetics has been successfully implemented and provides the framework for studies in the field of shock-induced combustion.

Key Results

- Diegelmann, F., Hickel S., Adams N.A., Shock Mach number influence on reaction wave types and mixing in reactive shock-bubble interaction. Combustion and Flame, 174, 085-099, 2016
- 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 316, 453-469, 2016
- Örley, F., Hickel, S., Schmidt, S.J., Adams, N.A., Large-eddy simulation of turbulent, cavitating flow inside a 9-hole diesel injector including needle movement. International Journal of Engine Research, 1-17, 2016
- Pasquariello, V., Hammerl, G., Örley, F., Hickel, S., Danowski, C., Popp, A., Wall, W.A., Adams, N.A., A cut-cell finite volume-finite element coupling approach for fluid-structure interaction in compressible flow. Journal of Computational Physics 307: 670-695, 2016
- Pasquariello, V., Grilli, M., Hickel, S., Adams, N.A., Large-eddy simulation of passive shock-wave/boundary-layer interaction control. International Journal of Heat and Fluid Flow, Vol. 49, 116-127, 2014

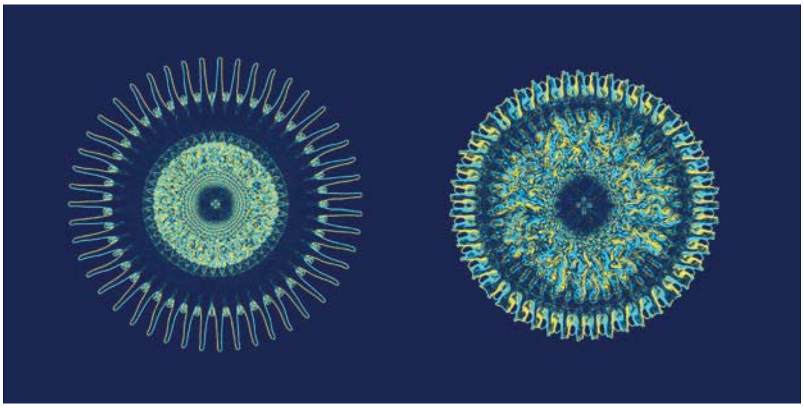
Multiscale Simulation Models for Complex Fluid Dynamics and Interactions

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 diverse fluids across interface segments. These interactions often can be described by local fluid properties. Simulations of multi-phase flow, such as bubble interaction, drop impact and spray atomization, need to resolve length scales that can span several orders of magnitude, which poses a great computational challenge. Adaptive mesh refinement (AMR) and multi-resolution (MR) methods, even with local time stepping, do not sufficiently reduce computational cost to enable accurate routine simulations of complex interfacial flows.

Approach to Solution

Based on our newly-developed multi-scale sharp interface modeling approach, we design numerical schemes with the ability to separate flow structure according to their characteristic length scales and to handle large scale and small scale structure in physically consistent ways. The developed numerical schemes achieve very good resolvability without compromising robustness. We have developed a high resolution scheme for transporting material interface networks and applied this scheme for the simulation of shock dynamics involving multiple compressible fluids.

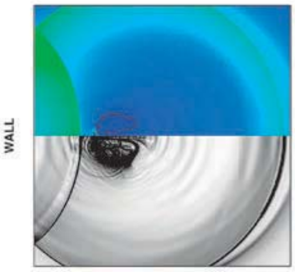


Multi-scale simulation of interface instability under an implosive shock

Key Results

- Efficient formulation of scale separation for multi-scale modeling of interfacial flows. J. Luo, X.Y. Hu, N.A. Adams. Journal of Computational Physics 308 (2016) 411-420.
- Curvature boundary condition for a moving contact line J. Luo, X.Y. Hu, N.A. Adams. Journal of Computational Physics 310 (2016) 329-341.
- On the convergence of the weakly compressible sharp-interface method for two-phase flows. S. Schranner, X.Y. Hu, N.A. Adams. Journal of Computational Physics 324 (2016) 94-114.
- A family of high-order targeted ENO schemes for compressible-fluid simulations. L. Fu, X.Y. Hu, N.A. Adams. Journal of Computational Physics 305, 333-359, 2015
- L.H. Han, X.Y. Hu, N.A. Adams, J. Comp. Phys. 262:131-152 (2014)
- L.H. Han, X.Y. Hu, N.A. Adams, J. Comp. Phys., 280:387-403 (2014)

NANOSHOCK* – Manufacturing Shock Interactions for Innovative Nanoscale Processes



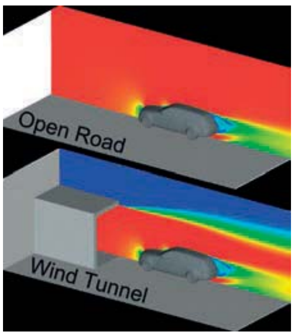
The figure shows an instantaneous simulation snapshot of a collapsing gas bubble near a solid wall. Due to the presence of the wall, the bubble collapse is asymmetrical and the wall reflects parts of the emitted shock wave (upper half: color map denotes pressure field, lower half: numerical schlieren image).

Since December 2015, the ‘Nanoshock’ group at the Institute of Aerodynamics and Fluid Mechanics (Prof. Adams) have been investigating the highly complex flow physics of shock interactions with interfaces. Shock waves are discontinuities in the macroscopic fluid state that can lead to extreme temperatures, pressures and concentrations of energy. A classic example of the generation of shock waves is the supersonic boom of an aircraft or the pressure wave originating from an explosion. Funded by the European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation program with an advanced grant for Prof. Adams, in this project we are developing numerical methods that are capable of predicting shock wave interactions with gas bubbles and soft tissue. Although frequently used applications such as kidney-stone lithotripsy rely on these phenomena, still today the underlying mechanisms are not fully understood. Using the most advanced numerical methods, we study the fundamental problem of gas-bubble collapse impact on interfaces and biomaterials and provide detailed quantitative data of the entire process. This knowledge can lead to improved treatments in cancer therapy or drug delivery by harnessing the enormous potential of shock-wave induced interactions.

N. Hoppe, J. Kaiser, A. Lunkov, V. Bogdanov, Dr.-Ing. S. Adami, Prof. Dr.-Ing. N. A. Adams

** 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).*

Automotive Aerodynamics



CFD simulation of DrivAer reference model in open road and wind tunnel conditions

Way to Solution

A new methodology to separate particular wind tunnel interferences is established on the base of CFD simulations using OpenFOAM®. The contribution of nozzle, open jet and collector interferences is evaluated by simulating the aerodynamics of the DrivAer reference model for both open road and wind tunnel conditions. Furthermore, the effect of moving ground simulation and its dimensions on race car aerodynamics are part of the CFD studies. The results from CFD are compared with classical blockage corrections. Finally, improved potential flow models are developed and design criteria for open jet test sections are derived.

Motivation

Wind tunnel testing is the essential method to develop automotive aerodynamics. Increasing nozzle size and test section length, as well as improved moving ground simulation, have been some of the most remarkable trends in the design of modern open jet facilities in the past decades. However, the typical blockage ratio is still in the range of 10% and hence open jet wind tunnel interferences are not negligible today. The requirement to increase accuracy goes along with a further development of open jet wind tunnel corrections.

Key Results

- Collin C., Mack S., Indinger T., Mueller J., ‘A numerical and experimental evaluation of open jet wind tunnel interferences using the DrivAer reference model,’ SAE Int. J. Passeng. Cars – Mech. Syst. 9(2):657-679, 2016, doi:10.4271/2016-01-1597.
- Collin C., Indinger T., Mueller J., ‘Moving ground simulation for high performance race cars in an automotive wind tunnel’, JSAE Annual Spring Congress 2016, Yokohama, Japan, 2015.

Aircraft and Helicopter Aerodynamics

Motivation

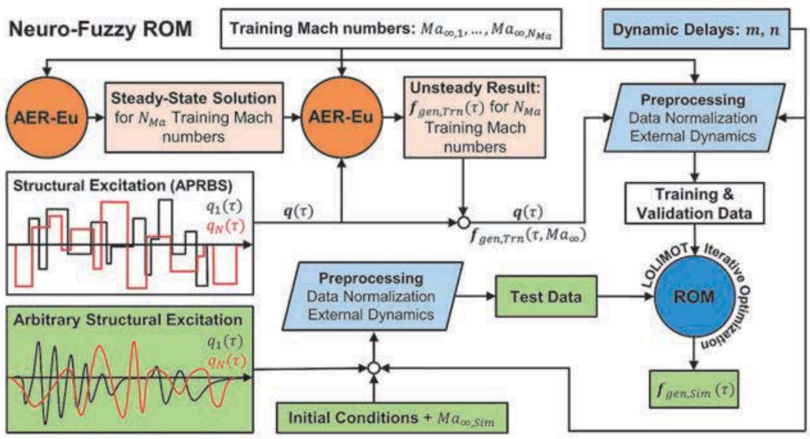
The long-term research agenda is based on the continued improvement of flow prediction and analysis capabilities in the context of aircraft and helicopter performance and drag reduction. Specific research activities are dedicated to the reliable prediction of flow separation onset and progression in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

Approach to Solution

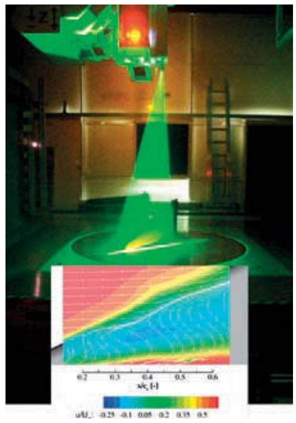
Investigations have been performed using both wind tunnel experiments and state-of-the art numerical simulations. In-house codes are continously further elaborated in the context of aeroelasticity analysis with respect to time-accurate fully coupled simulations as well as the application of novel neuro-fuzzy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

Key Results

- A. Hövelmann, F. Knoth, and C. Breitsamter. Aerospace Science and Technology, Vol. 57, 2016, pp. 18-30.
- A. Hövelmann, M. Grawunder, A. Buzica, and C. Breitsamter. Aerospace Science and Technology, Vol. 57, 2016, pp. 31-42.
- M. Grawunder, R. Reß, and C. Breitsamter. AIAA Journal, Vol. 54, No. 6, 2016, pp. 2011-2015.
- M. Grawunder, R. Reß, V. Stein, C. Breitsamter, and N. A. Adams. NNFM, Vol. 132, Springer Verlag, 2016, pp. 303-313.
- K. Kato, C. Breitsamter, and S. Obi. International Journal of Heat and Fluid Flow, Vol. 61, 2016, pp. 58-67.
- A. Kölzsch, S. Blanchard, and C. Breitsamter: NNFM, Vol. 132, Springer Verlag, 2016, pp. 823-832.
- M. Winter, and C. Breitsamter. AIAA Journal, Vol. 54, No. 9, 2016, pp. 2705-2720.
- M. Winter, and C. Breitsamter. Journal of Fluids and Structures, Vol. 67, 2016, pp. 2705- 2720.
- J. H. You, C. Breitsamter, and R. Heger. CEAS Aeronautical Journal, Vol. 7, No. 2, 2016, pp. 185-207.

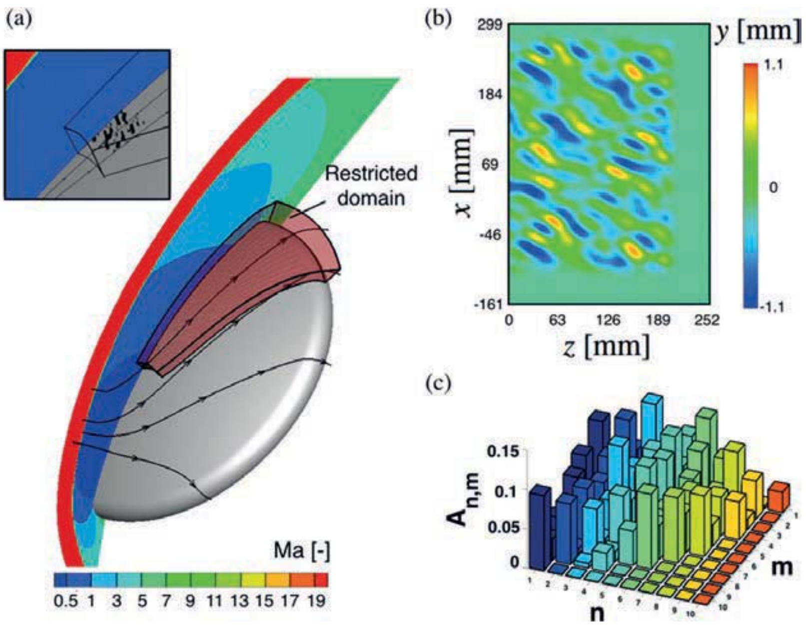


Structure of neuro-fuzzy ROM chain for aeroelastic analysis



Stereo-PIV measurements on controlled delta wing flow

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



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

Motivation

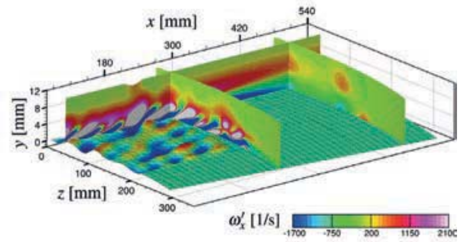
Blunt bodies returning from space are subject to immense heat loads leading to ablation. Roughnesses on these ablating surfaces can induce laminar-turbulent transition, which again increases the heat load on the surface. This self-energizing effect can lead to a catastrophic failure of the spacecraft. The role of 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: High-temperature gas effects on the high-enthalpy flow on the windward-side boundary layer of a generic capsule geometry. 8th ESA Symposium on Aerothermodynamics for Space Vehicles; Lisbon, 2015.
- A. Di Giovanni, C. Stemmer, Numerical simulations of the high-enthalpy boundary layer on a generic capsule geometry with roughness, DGLR-STAB Symposium Braunschweig, 2016.



DNS results from vortical disturbances induced by surface roughness in a reacting environment

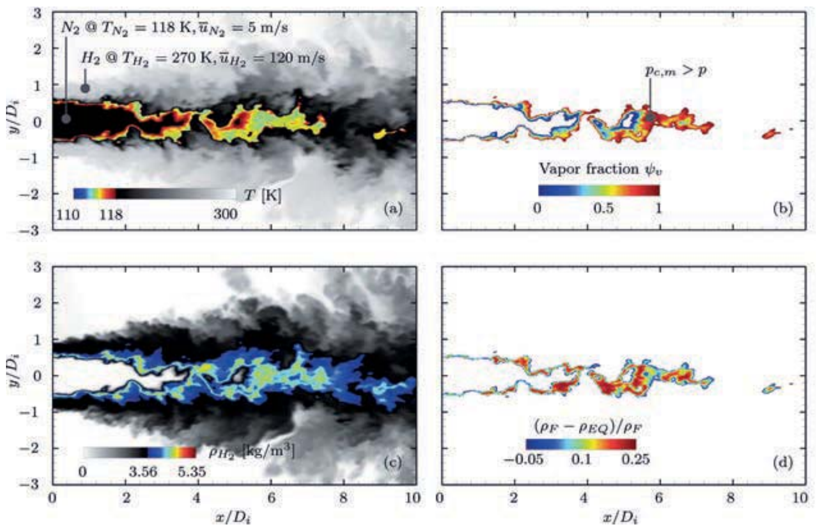
SFBTR40 DFG Sonderforschungsbereich TRR 40: Technological Foundations for the Design of Thermally and Mechanically Highly Loaded Components of Future Space Transportation Systems

The institute has the speaker role within the DFG-SFB TRR 40. 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 TRR 40 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 an increase in efficiency 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 developed concepts 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 TRR 40.
The scientific focus of all five research areas within the TRR 40 is the analysis and 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.

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



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

Courses

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

Management

Prof. Dr.-Ing. Nikolaus A. Adams, Director
Apl. Prof. Dr.-Ing. Habil.
Christian Breisamter
PD Dr.-Ing. Habil. Christian Stemmer
Dr.-Ing. Xiangyu Hu
PD Dr.-Ing. Habil. Thomas Indinger

Adjunct Professor

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
Dmitrii Azarnykh, M.Sc.
Bruno Beban, M.Sc.
Dr.-Ing. Xin Bian
Vladimir Bogdanov, M.Sc.
Morgane Borreguero, M.Sc.
Dipl.-Ing. Bernd Budich
Dipl.-Ing. Andrei Buzica
Yuanwei Cao, M.Sc.
Michael Cerny, M.Sc.
Dipl.-Ing. Christopher Collin
Antonio Di Giovanni, M.Sc.
Felix Diegelmann, M.Sc.
Alexander Döhring, M.Sc.
Zheng Fang, M.Sc.
Lin Fu, M.Sc.
Dr.-Ing. Marcus Giglmaier
Polina Gorkh, M.Sc.
Dipl.-Ing. Lukas Haag
Dr.-Ing. Luhui Han
Nils Hoppe, M.Sc.
Zhe Ji, M.Sc.
Jakob Kaiser, M.Sc.
Dipl.-Ing. Thomas Kaller
Marco Kiewat, M.Sc.
Dipl.-Ing. Florian Knoth
Aleksandr Lunkov, M.Sc.
Xiuxiu Lyu, M.Sc.
Dipl.-Ing. Jan Matheis
Daiki Matsumoto, M.Sc.
Lu Miao, M.Sc.
Dipl.-Phys. Christoph Niedermeier
Daria Ogloblina, M.Sc.
Aleksandra Pachalieva, M.Sc.
Shucheng Pan, M.Sc.
Dipl.-Ing. Vito Pasquariello
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. Felix Schranner
Dipl.-Ing. Victor Stein
Dipl.-Ing. Marco Stuhlpfarrer
Theresa Trummler, M.Sc.
Konstantin Vachnadze, M.Sc.
Jianhang Wang, M.Sc.
Zhaoguang Wang, M.Sc.
Dipl.-Ing. Maximilian Winter
Chi Zhang, M.Sc.

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

Publications 2016

- Azarnykh, D., Litvinov, S., Bian, X., Adams, N.A: Determination of macroscopic transport coefficients of a dissipative particle dynamics solvent; Physical Review E – Statistical, Nonlinear, and Soft Matter Physics; Volume 93, Issue 1, 11 January 2016, article number 013302; DOI: 10.1103/PhysRevE.93.013302
- Azarnykh, D., Litvinov, S., Adams, N.A.: Numerical methods for the weakly compressible generalized Langevin model in Eulerian reference frame; Journal of Computational Physics; Volume 314, June 01, 2016, pp. 93-106; DOI: 10.1016/j.jcp.2016.02.073
- Bartasevicius, J., Buzica, A., Breitsamter, C.: Discrete vortices on delta wings with unsteady leading-edge blowing; 8th AIAA Flow Control Conference 2016; Washington; United States; 13 June 2016-17 June 2016; code 175889
- Dillmann, A., Heller, G., Krämer, E., Wagner, C., Breitsamter, C.: New results in numerical and experimental fluid mechanics X: Contributions to the 19th STAB/DGLR symposium Munich, Germany, 2014; Notes on Numerical Fluid Mechanics and Multidisciplinary Design; Volume 132, 2016; DOI: 10.1007/978-3-319-27279-5
- Dillmann, A., Heller, G., Krämer, E., Wagner, C., Breitsamter, C.: Preface; Notes on Numerical Fluid Mechanics and Multidisciplinary Design; Volume 132, 2016, pp. v-vii
- 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, July 01, 2016, pp. 453-469; DOI: 10.1016/j.jcp.2016.04.021
- Fu, L., Hu, X.Y., Adams, N.A.: A family of high-order targeted ENO schemes for compressible-fluid simulations; Journal of Computational Physics; Volume 305, January 15, 2016; DOI: 10.1016/j.jcp.2015.10.037
- Grawunder, M., Reß, R., Stein, V., Breitsamter, C., Adams, N.A.: Validation of a flow simulation for a helicopter fuselage including a rotating rotor head; Notes on Numerical Fluid Mechanics and Multidisciplinary Design; Volume 132, 2016, pp. 303-313; DOI: 10.1007/978-3-319-27279-5_27
- Hu, X.Y.: Simple gradient-based error-diffusion method; Journal of Electronic Imaging, DOI: 10.1117/1.JEI.25.4.043029; Volume 25, Issue 4, 1 July 2016, article number 043029
- Kölzsch, A., Blanchard, S., Breitsamter, C.: Dynamic actuation for delta wing post stall flow control; Notes on Numerical Fluid Mechanics and Multidisciplinary Design; Volume 132, 2016, pp. 823-832; DOI: 10.1007/978-3-319-27279-5_72
- Luo, J., Hu, X.Y. , Adams, N.A.: Curvature boundary condition for a moving contact line; Journal of Computational Physics; Volume 310, April 01, 2016, pp. 329-341; DOI: 10.1016/j.jcp.2016.01.024
- Luo, J., Hu, X.Y. , Adams, N.A.: Efficient formulation of scale separation for multi-scale modeling of interfacial flows; Journal of Computational Physics; Volume 308, March 01, 2016, pp. 411-420; DOI: 10.1016/j.jcp.2015.11.044
- Pasquariello, V., Hammerl, G., Örley, F., Hickel, S., Danowski, C., Popp, A., Wall, W.A., Adams, N.A.: A cut-cell finite volume – finite element coupling approach for fluid-structure interaction in compressible flow; Journal of Computational Physics; Volume 307, February 15, 2016, pp. 670-695; DOI: 10.1016/j.jcp.2015.12.013
- Schranner, F.S. , Hu, X. , Adams, N.A.: On the convergence of the weakly compressible sharp-interface method for two-phase flows; Journal of Computational Physics; Volume 324, 1 November 2016, pp. 94-114
- Schranner, F.S., Rozov, V., Adams, N.A.: Optimization of an implicit large-eddy simulation method for underresolved incompressible flow simulations; AIAA Journal; Volume 54, Issue 5, 2016, pp. 1567-1577; DOI: 10.2514/1.J054741
- Winter, M., Breitsamter, C.: Efficient unsteady aerodynamic loads prediction based on nonlinear system identification and proper orthogonal decomposition; Journal of Fluids and Structures; Volume 67, 2016, pp. 1-21; DOI: 10.1016/j.jfluidstructs.2016.08.009
- Winter, M., Breitsamter, C.: Neurofuzzy-model-based unsteady aerodynamic computations across varying freestream conditions; AIAA Journal; Volume 54, Issue 9, 2016, pp. 2705-2720; DOI: 10.2514/1.J054892
- Winter, M., Breitsamter, C.: Efficient modeling of generalized aerodynamic forces across Mach regimes using neuro-fuzzy approaches; Notes on Numerical Fluid Mechanics and Multidisciplinary Design; Volume 132, 2016, pp. 467-477; DOI: 10.1007/978-3-319-27279-5_41
- You, J.H., Breitsamter, C., Heger, R.: Numerical investigations of Fenestron™ noise characteristics using a hybrid method; CEAS Aeronautical Journal; Volume 7, Issue 2, 1 June 2016, pp. 185-207; DOI: 10.1007/s13272-015-0180-1
- Zwerger, C., Hickel, S., Breitsamter, C., Adams, N.: Wall modeled large eddy simulation of a delta wing with round leading edge; Notes on Numerical Fluid Mechanics and Multidisciplinary Design; Volume 132, 2016, pp. 607-616; DOI: 10.1007/978-3-319-27279-5_53

Flow Control and Aeroacoustics

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

The focus of the research group in 2016 was the development and testing of research tools for the numerical prediction of flow and sound fields and for their validation in wind-tunnel experiments.

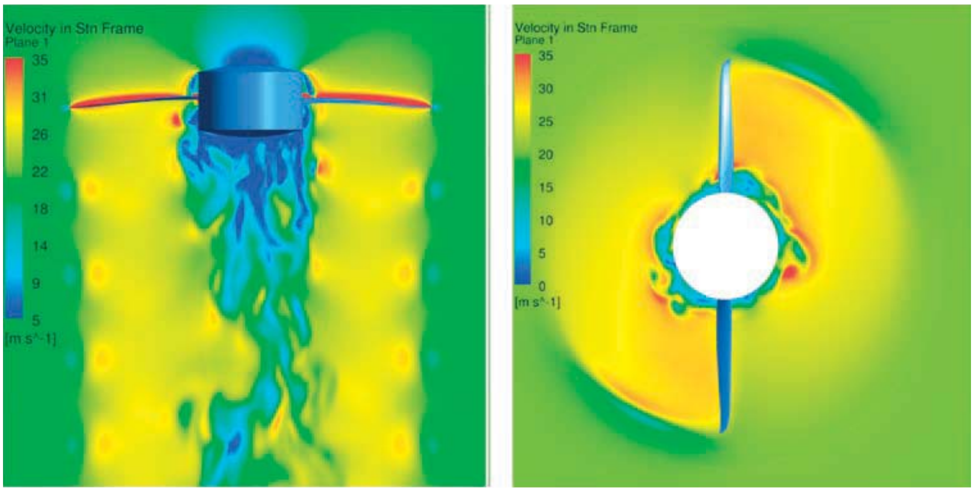
Prof. Dr.-Ing.
Hans-Jakob Kaltenbach

Contact

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

Sound Radiation from a 2-bladed Pusher Propeller

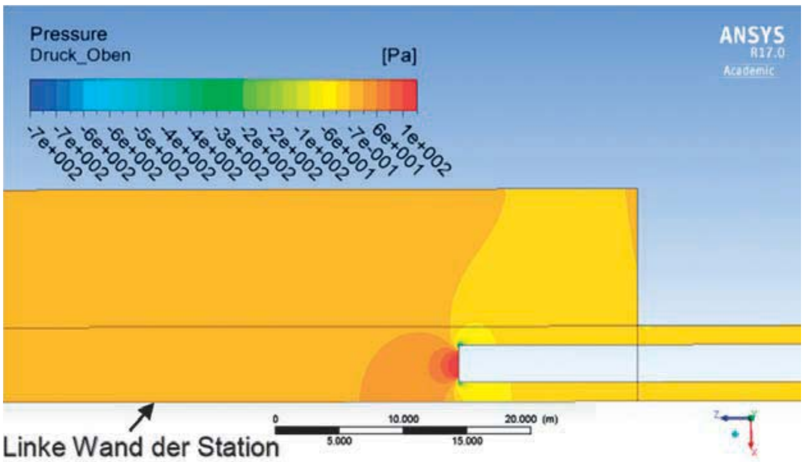
As a step towards better understanding of installation effects on aerodynamic noise generation, a pusher propeller configuration is investigated. It can be installed downstream of the V-shaped tail section of an UAV (corresponding to the IMPULSS geometry designed and built at LLS under the auspices of Prof. M. Hornung). The hybrid prediction approach consists of two steps. Firstly, an unsteady CFD simulation employing sliding-mesh technique is carried out in the time domain, yielding unsteady pressure on the blade surface. In a second step, the Ffowcs-Williams Hawkings solver Spysi (developed at the Friedrich-Alexander University of Erlangen in the group of Prof. S. Becker) is employed to predict the far-field sound field.



Isosurface of the magnitude of the velocity vector in a cut along the propeller axis (left) and through the propeller plane (right). (from M. Habryka, student thesis)

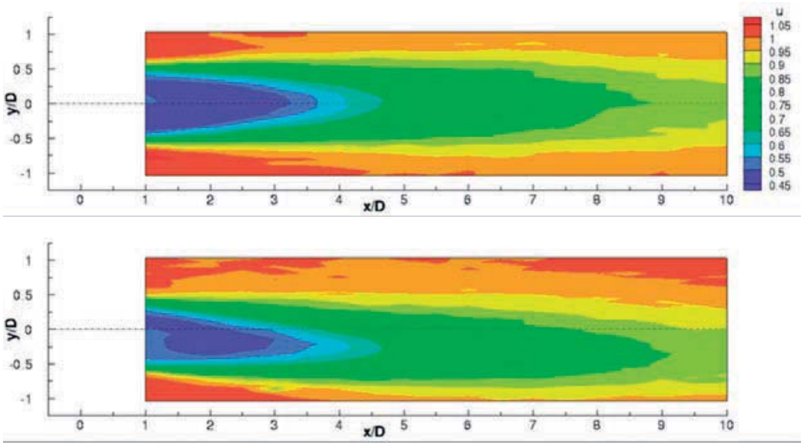
Numerical Prediction of Aerodynamic Loads During Entry of a Subway Train into an Underground Station

The wall cladding and other installations in subway stations experience unsteady aerodynamic loads during entry and passage of trains. The purpose of an ongoing project is the quantitative prediction of these loads by numerical simulation (CFD) and possible verification by measurements at real sites at a later stage. Up to now, results from transient simulations based on the sliding-mesh technique have been obtained for generic train and station geometries at parameters (train speed, deceleration, blockage, distance of stations) corresponding to a typical situation in the Munich subway system.



Isosurface of the pressure field on a horizontal cut in 2m distance from the ground of the subway station at an instance of time, when the train head has already entered the station (from T. S. Klose, B.Sc. thesis, 2016)

Measurement of Wake Development of a Model Wind Turbine over Rough Terrain



Axial component of the mean velocity in a horizontal plane through the rotor hub for normal orientation (top) of the rotor and for a yawed configuration (bottom) (from Ch. Zhang, M.Sc. thesis, 2016)

Prediction of the evolution of the wakes forming behind wind turbines is a necessary ingredient for the optimization of wind farms with respect to maximum power production and load alleviation. Experiments on a model-scale turbine with a rotor diameter of 450 mm have been carried out in the boundary layer wind tunnel of the TUM Chair of Aerodynamics. The influence of surface roughness (terrain conditions), tip-speed ratio and yaw angle on the wake evolution is studied. Detailed flow field measurements have been carried out with hot-wire probes.

Projects

IGSSE project EUROTECH-Greentech 03-Wind.

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 setup of a microphone array

Courses

- Continuum Mechanics (MSE), 50 %
- Grundlagen der numerischen Strömungsmechanik
- Aeroakustik
- Strömungsbeeinflussung
- Numerische Strömungsakustik
- Praktikum Numerische Strömungssimulation
- Praktikum Numerische Strömungsakustik

Management

Prof. Dr.-Ing. Hans-Jakob Kaltenbach

Administrative Staff

Amely Schwörer (secretary to the Chair of Aerodynamics)

Research Scientists

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

Publications 2016

- Stein, Victor P., Kaltenbach, H.-J.: Wind-tunnel modelling of the tip-speed ratio influence on the wake evolution. Journal of Physics: Conference Series 753 (2016) 032061