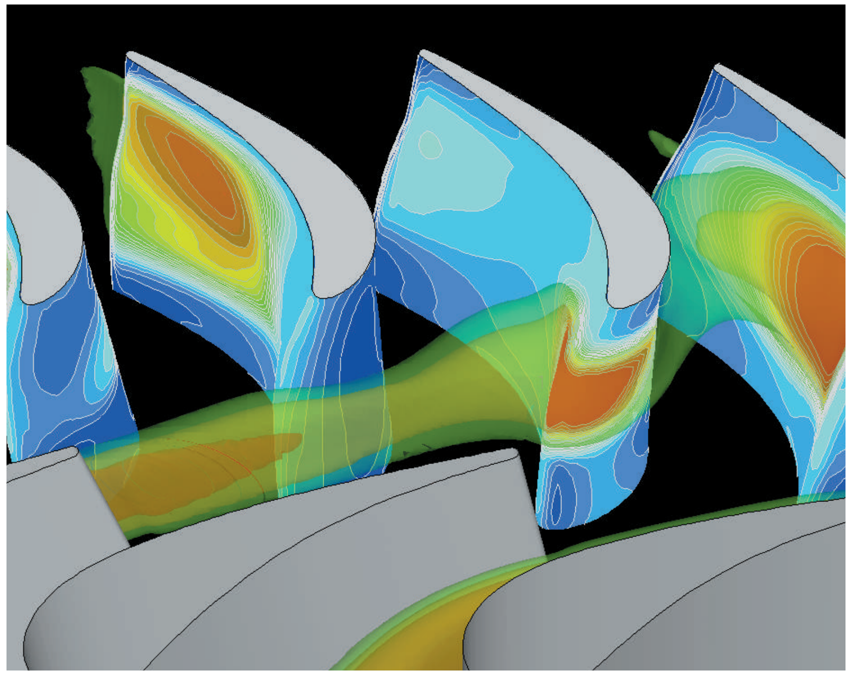


Focus of Future Development

The already sophisticated simulation system TRACE continues to be developed further by more than ten scientists within the framework of internal, national and European research projects. The close cooperation with the industrial partners ensures the orientation of development remains relevant to the design driven demands of industry. An important focus of research is therefore turbulence and transition modelling, with consideration of unsteady three-dimensional effects as well as efficiency and roughness for industrial applications. For the calculation of complex geometries TRACE has been extended to an unstructured solver based on the validated numerical techniques of the structured solver. The coupling of structured and unstructured grids simplifies grid generation and reduces costs for complex geometries, e.g. case contouring or cavities. Furthermore, interdisciplinary problems of aeroelasticity, aeroacoustics and thermodynamics are of increasing importance. For thermodynamic problems, coupled fluidstructure calculations enable predictions of the heat transfer on blade surfaces. An example of a current numerical study is the analysis of the thermal loading of a high-pressure turbine stage with consideration of hot gas streaks. Alongside the further development of physical models, the development of improved numerical methods, such as new temporal integration methods to reduce computation overheads, is important. Within the development framework of TRACE software engineering plays an important role: the complete simulation process chain is integrated into a version management system, including the pre- and post-processing tools, with a standard interface (CGNS). In summary, the TRACE program has become a highly accurate and capable CFD platform for modelling turbo-machinery related flow phenomena.

Figure: thermic loading of a high-pressure turbine stage caused by hot gas streaks.

Contact :

DLR - German
Aerospace Center

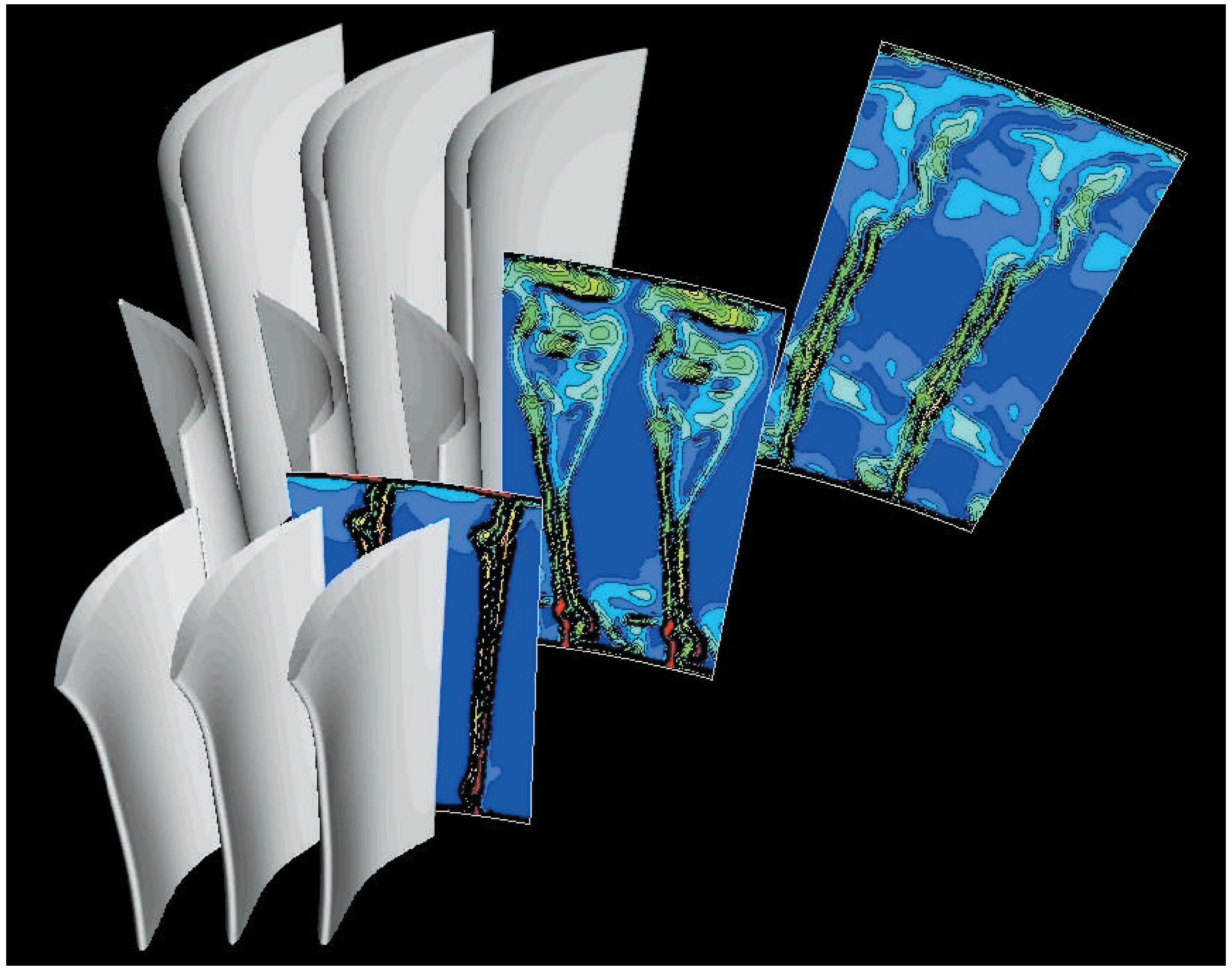
Institute of Propulsion Technology
Linder Höhe
51147 Cologne

Phone: 0 22 03/ 6 01-45 75
Fax: 0 22 03/ 6 43 95

E-Mail: Dirk.Nuernberger@dlr.de
Edmund.Kuegeler@dlr.de

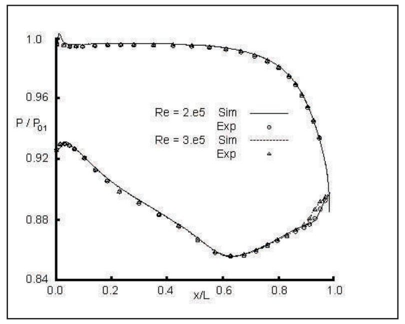
http://www.trace-portal.de

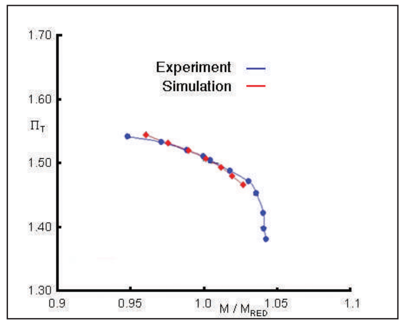
TRACE-
Modern Simulation Technologies for Turbomachinery Design



In turbomachinery design there is a clear trend towards shortened development cycles. Following the development of powerful computing resources, numerical simulation technologies now represent an indispensable tool within the development cycle. At the Institute for Propulsion Technology of the German Aerospace Center (DLR) a simulation system has been developed specifically for the calculation and investigation of turbomachinery flows. After a decade of development and validation the simulation system has reached maturity under the name TRACE (Turbo machinery Research Aerodynamic Computational Environment) and become established as the standard method for the computation of internal flows within DLR. The program has been successfully used at research and university institutes for the scientific analysis of complex, unsteady turbomachinery flows. Furthermore, the system has been applied in industry for several years in the development and optimization of engine components. As a result of the close cooperation with MTU-Aeroengines, industrial requirements concerning data handling and efficiency have helped shape the development of the system.

Figure: turbulent loss areas of a low pressure turbine within a real aero-engine.





Development Status

From the beginning TRACE has been developed specifically to investigate the highly complex time-dependent turbo-machinery flows. Over time, a team of scientists has developed a sophisticated simulation system with high quality numerical models. TRACE is a structured multi-block flow solver based on the fully compressible Navier-Stokes equations. An upwind spatial discretization scheme combined with an implicit time integration method guarantees, alongside efficiency and robustness, a highly accurate solution. Non-reflecting boundary conditions and fully conservative interpolation techniques enable a realistic reproduction of physical boundary conditions as well as a flexible coupling of rotating and non-rotating grid blocks.

Figure upper left: density gradient in a stator-rotor configuration of a low-pressure compressor.

Figure upper right: pressure distribution on the blade surface of a turbine cascade: experiment-simulation.

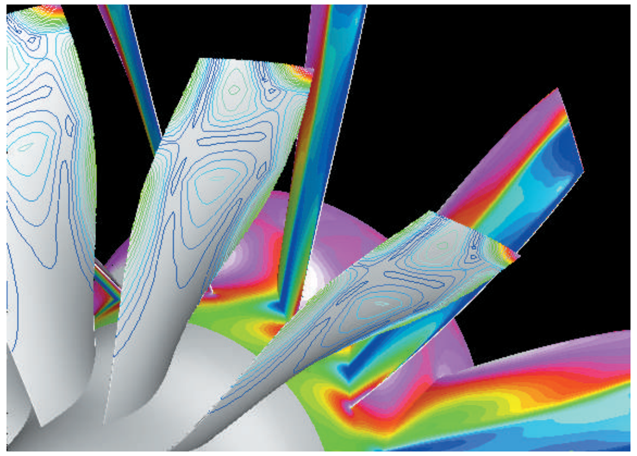
Figure lower right: performance map of a transsonic compressor stage: experiment-simulation

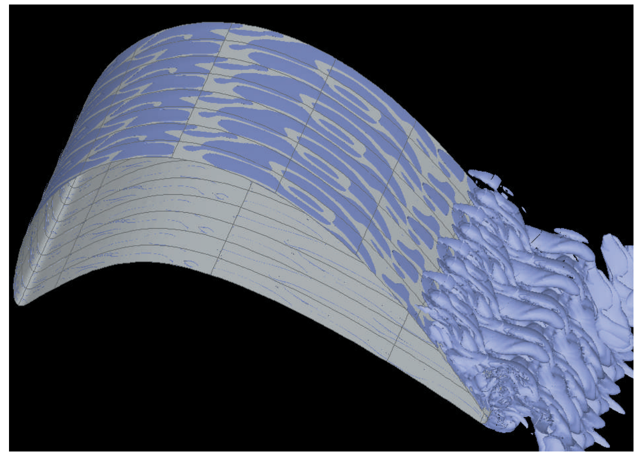
To account for the effects of turbulence, both the Wilcox k-omega model, and an extended version of the well known Spalart and Allmaras model are available. Furthermore, transition criteria coupled directly with the turbulence models serve to detect the transient laminar/turbulent junctions common in many turbomachinery flows. The system is written in the C programming language and is embedded in a parallel environment. The code is hardware independent and may therefore be run on any processor architecture. A very efficient parallel scalability is reached on machines ranging from simple PC clusters to massively parallel super-computers with several hundred processors.

To establish confidence in a CFD program requires extensive validation with existing computational and experimental data. Throughout TRACE's development the code has been continuously validated against numerous test cases. When com-paring experimental and computational data a commonly used measure in turbo-machinery applications is the distribution of static pressure over a blade surface. The excellent agreement in the data, and in particular the accurate prediction of the separation bubble at low Reynolds number, and laminar-turbulent transition by bypass-transition at higher Reynolds number, verify the high quality of the simulation system TRACE. The operating behaviour of multi-stage turbomachines can be estimated through integral parameters such as pressure rise and flow rate. Good agreement between the measured and computed characteristic curves of a given turbomachine indicates that the three-dimensional flow-phenom-ena are captured to sufficient accuracy in the numerical simulation

Flow Solutions

On account of component reduction and greater thrust requirements, the aerodynamic and mechanical requirements of modern engines have risen dramatically. As a consequence of the trend towards compact designs, e.g. through the reduction of the axial-gap between adjacent blade rows, the significance of the interaction between blade rows has grown; and therefore a detailed knowledge of these phenomena is now of great importance in the optimization of blade design. The understanding of these phenomena is a task ideally suited to the TRACE system. As an example of the level of accuracy achievable, the results of a time-accurate compressor simulation are shown in terms of instantaneous density gradients (numerical Schlieren).





The rotor shock interacts with the up-stream stator inducing strong aero-dynamic and structural forces. An accurate prediction of theefficiency and losses can only be achieved by accurately resolving these non-linear effects.

Coupled Applications

As methods are progressively refined, interdisciplinary questions become of ever greater importance. To determine the properties of materials for instance, know-ledge of the thermal blade loading is necessary. This can only be obtained through a coupled calculation of the outer flow and the heat conduction within the blade. In a pilot project the TRACE code was successfully coupled with a heat conduction process through a universal interface. Another interdisciplinary area of importance is aeroelasticity. Through the interaction between the flow and the blades, the danger (particularly in heavily loaded blade rows) of exciting blade vibrations exists. Operating conditions with large amplitude oscillations need to be excluded in the design phase as they may lead to blade failure in the extreme. For an accurate prediction of the aeroelastic interactions the simulation system TRACE has a module for the inclusion of blade deformation. It has been developed in cooperation with the DLR Institute of Aeroelasticity in Göttingen. The unsteady aerodynamics and the structural dynamics of the blades are calculated simultaneously in a coupled simulation. Different aeroelastic problems can be considered: aerodynamically self-excited blade vibration (flutter) as well as blade vibration excited by an external aerodynamic perturbation (forced response). Current studies include forced response calculations of a counter rotating propfan. The excitation of the front rotor, which is caused by the pressure field of the rear rotor, can be determined by analyzing the time-accurate simulation data.

Acoustic Solutions

Engine noise is a topical research problem of considerable importance. Today, the radiation of discrete frequency sound, tonal noise, can already be calculated with the appropriate numerical methods. To accurately capture the sound waves, the amplitudes of which are three to five orders of magnitude below atmospheric pressure, the TRACE code is being developed further to incorporate high-order accurate spatial and temporal numerical methods. With consideration of the necessary resolution, the first calculations have been successfully performed. The noise spectrum includes both discrete frequency tonal and stochastic components, the latter being known as broadband noise. A promising method for the calculation of broadband noise is the Large Eddy Simulation method. In this approach the larger turbulent scales are directly resolved. Simplifying assumptions about turbulence are avoided. The finer turbulence structures not resolved are taken into account through so called sub-grid scale models. The results of such LES simulations are important in understanding the formation mechanisms of the large swirl structures that often occur in the rear blade area of turbines.

Figure upper left: "Forced-Response"-simulation of a counter rotating propfan. Aerolastic excitation of front rotor by pressure field of rear rotor.

Figure upper right: vortex structures near the suction surface of a low pressure turbine as a result of a LES-simulation.