The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Complex Flows

Y. Egorov¹, F. Menter¹, R. Lechner¹ and D. Cokljat²

 ¹ ANSYS Germany GmbH, Staudenfeldweg 12, 83624 Otterfing, Germany
 ² ANSYS UK Ltd, Sheffield Business Park, 6 Europa View, Sheffield, S9 1XH, United Kingdom
 Phone: +49(0)8024 9054 15
 Fax: +49(0)8024 9054 17
 E-mail: Florian.Menter@ansys.com
 www.ansys.com

The paper summarises the validation activity performed with the Scale-Adaptive Simulation model of turbulence (SAS model) using the two commercial CFD solvers, namely ANSYS-FLUENT and ANSYS-CFX. Both the KSKL-SAS and the SST-SAS model variants have been tested, although major experience is obtained with the latter one. The turbulence-resolving capability of the SAS method has been validated with a representative set of test cases, covering both underlying generic flows as well as practical engineering applications. Most of the test case simulations were conducted during the recent EU project "DESider". In addition to the purely aerodynamic flows with massive separations and heat transfer they include also such physical phenomena as turbulent combustion and aeroacoustics. The illustrating results show the potentials of the SAS approach for industrial flow simulations.

Keywords: turbulence model; scale-adaptive simulation; SAS; hybrid RANS-LES; separated flow; aero-acoustics

Introduction

In a companion article (Menter and Egorov, [15]), the theory and rational behind the Scale-Adaptive Simulation (SAS) methodology is given in detail. However, the testcases in that article where mainly restricted to generic flows suitable for demonstrating the basic behaviour of the SAS concept. The current article aims at providing a wider range of testcases, with an emphasis on industrial applications; it should be read in combination with Menter and Egorov [15]. It should also be noted that SAS is not suitable for all unsteady simulations, and the limitations of the formulation are discussed in detail in Menter and Egorov [15]. The current contribution focuses therefore on successful applications of the method to outline the type of flows suitable for SAS model applications. These are mainly flows featuring strong flow instabilities typically associated with large separated zones behind bluff bodies or flows with vortical instabilities.

The two SAS models described in Menter and Egorov [15] were implemented into the two commercial CFD solvers, ANSYS-CFX and ANSYS-FLUENT (KSKL-SAS – in an experimental version of the CFX solver only), and applied to a number of simple to complex test cases. Both solvers are based on control volume methods for unstructured grids. FLUENT uses the cell-centred polyhedral approach, while CFX builds the dual-mesh control volumes around the grid vertices. The reason for using the two solvers with different discretisation techniques was to prove the model versatility and code-independence, which is of particular importance for LES/DES/SAS type of simulations.

The testcases presented here demonstrate the performance of the method for the prediction of wide range of practical flows. They include external aerodynamic flows (a 2-D airfoil at high incidence and a full aircraft configuration), as well as internal flows with heat and mass transfer (turbine blade cooling, swirl burner, hot buoyant jet in cross flow). Besides, two test cases deal with aeroacoustics. The external aerodynamics and the aeroacoustics test cases were the official test cases of the DESider project (Haase et al., [6]).

It should be noted that the SAS model is already widely used in industrial flow simulations, with applications going far beyond the current generic cases. Simulations include flows:

- in combustion chambers (Widenhorn et al. [23])
- around turbine blades (Joo and Durbin, [11]),
- in rotating cavities of gas turbines (Smirnov et al. [21]),
- in chemical mixers (Fletcher et al. 2007, Honkanen et al. [9]),
- with unsteady thermal loading in nuclear reactor components (Frank et al.
 [4])
- Francis runner (Magnoli and Schilling, [13])
- car mirrors (Grahs and Othmer, [5])
- generic internal combustion engine (Imberdis et al.[10])

as well as many unpublished applications.

Internal flows with heat and mass transfer, buoyancy, combustion

Turbine blade cooling

One of the limiting factors in the design of high-pressure turbine blades is the maximum temperature on the blade surface. In order to increase the overall efficiency, the first blades behind the combustion chamber are equipped with active cooling devices. The current application is for the film cooling of the trailing edge of a turbine blade. Due to the decreasing thickness of the blade near the trailing edge, the cooling is achieved by a cooling film injected parallel to the blade surface. Figure 1 shows the geometry of the blade and the surface grid for the AITEB testcase by Martini et al. [14]. There are two inlet regions for this simulation. On the upper inlet, the hot gas enters the domain and at the inlet to the cooling channel, cold gas is injected. The cold gas does however pass over a hot wall before it reaches the mixing zone. It does therefore not stay at the inlet temperature. The reference temperature for the cold gas is taken downstream of the cold gas inlet. It is therefore not the value of the cold gas at the inlet. In the simulations, the reference temperature was taken at the same location as in the

experiment. The upper boundary of the domain is a free slip adiabatic wall. Periodicity is applied at the side planes. This testcase is courtesy of Dr. Lutum of MTU Aero Engines and has been investigated within the EU-project AITEB, G4RD-CT-1999-00055.

The finite-volume grid consists of $6.48 \cdot 10^5$ hexahedral elements. The walls are resolved with the $y^+ \sim 1$. A time step of $\Delta t=0.01 \cdot 10^{-3}$ s was used. (Characteristic velocity 50 m/s and dimension $L\sim 0.1$ m).

The main parameter for the evaluation of the device is the cooling efficiency. It is defined as follows:

$$\eta = \frac{T_{in}^{hot} - T_w}{T_{in}^{hot} - T_{ref}^{cold}}$$

where T_{in}^{hot} is the temperature of the hot gas at the inlet, T_w is the computed wall temperature at the adiabatic wall section, and T_{ref}^{cold} is the reference low temperature taken at the reference point shown in Figure 1.

Figure 2 shows the cooling efficiency (averaged in time and spanwise direction) for three different simulations against the experimental data. Clearly, both steady RANS and URANS do not provide sufficient mixing to reproduce the experimental results. The cooling efficiency is therefore computed too optimistic, as the trailing edge surface is shielded from the hot gas. The SAS model produces a significantly stronger mixing of the two streams and results in a much better agreement of the trends of the cooling efficiency with the experiments.



Figure 1: Schematic of the computational domain and the location of reference point for the cooling temperature

Figure 2: Cooling efficiency for different versions of the SST model compared to experimental data

Figure 2 shows deviations between all three simulations and the experiments at the start of the adiabatic wall. This might not be so much a deficiency in the simulations in the mixing zone, but a systematic mismatch with the reference conditions of the experiment. The most likely reason is a difference in the reference temperature T_{ref}^{cold} due to different inlet conditions of the cold stream, or an incorrect prediction of the heat transfer upstream of the reference point. Nevertheless, the results demonstrate the improved performance of the SAS model vs. standard URANS or RANS simulations, assuming that the shift between SAS and experiments is mainly a result of the difference in reference temperature. For a similar application showing a systematic improvement of SAS vs. URANS, see Joo and Durbin [11].

Figure 3 shows the turbulent structures computed by the SAS model. They represent an iso-surface of Ω^2 -S²=10⁵ 1/s². The colour represents the turbulent length scale vs. the height of the base, H (see Figure 1). The strong mixing zone behind the bluff body of the divider between hot and cold gas can be clearly seen. The unsteady mixing is responsible for the increase in heat transfer between the hot and the cold gas. This is a typical example for the application of the SAS model in technical flows. Automatically, the attached boundary layers are covered by steady RANS and the problematic mixing zone is resolved in scale-resolving mode by the SAS-part of the model.

Figure 3: Turbulent structures computed by the SST-SAS model for a film cooling test case

Hot buoyant jet in cross flow in a channel

This test case has been designed and studied at the Institute of Fluid Dynamics, ETH Zurich, in the framework of the EU project "Cost Action C17, Built Heritage: Fire loss to Historic Buildings". It is aimed at the experimental and numerical investigation of mixed convective flow with heat and mass transfer during fires in confined spaces.

Figure 4: Experimental installation for the hot buoyant jet in cross flow.

The experimental setup is shown in Figure 4. A long horizontal channel of a square cross section of $0.8 \text{ m} \times 0.8 \text{ m}$ is ventilated by air at room temperature with the mean flow velocity of 0.73 m/s. A hot air jet at 500°C is injected from below through a vertical circular pipe of 0.2 m diameter with the mean flow velocity of

2.8 m/s. More details about the experiment are provided by (Rusch, [17]). Figure 5 shows the unsteady flow structures computed using the current KSKL-SAS model of ANSYS-CFX. A hexahedral grid with $1.2 \cdot 10^6$ elements was used in the simulations, the time step was $\Delta t=2 \cdot 10^{-3}$ s. Around 4000 time steps (8 convective time units) have been computed to establish the stratified flow before starting the averaging procedure and the additional 6000 time steps – to obtain the averaged results.

Figure 6 shows the temperature profiles downstream of the injection location in the middle of the channel. The measured and the calculated averaged profiles are plotted along the vertical lines on the middle of the channel, on cross sections located 2 m, 4 m and 6 m downstream from the injection pipe axis. Steady state RANS simulations using the SST model clearly miss the mixing of the hot jet with the ambient air from the inlet. Superior agreement with the experiment is achieved by using the KSKL-SAS model due to the enhanced mixing in the unsteady region downstream of the jet.

Figure 5: Turbulent structures for hot jet flow (iso-surface of $Q=S^2-W^2$)

Figure 6: Profiles of the averaged temperature at the central plane. Distance from the hot inlet axis: 1.5 m, 3.5 m, and 5.5 m from left to right.

Turbulent combustion in a swirl burner

The instabilities in gas turbine combustion chambers are of strong technical interest, as they can produce noise and also compromise the structural integrity of the chamber. The unsteadiness can be caused by different mechanisms like flow instabilities introduced by the high swirl in the burner or thermo-acoustic instabilities from combustion itself.

The SST-SAS model is applied to the flow in a single swirl burner investigated experimentally by Schildmacher and Koch (Schildmacher et al., [18]) at ITS (Institut für Thermische Strömungsmaschinen) of the University of Karlsruhe. The ITS burner is typical for industrial gas turbine combustion systems. The test rig was built as a rectangular combustion chamber, the installation details are described in (Schildmacher, [18]). A lean pre-heated methane-air mixture is supplied through a ring inlet with the external diameter of 120 mm, which encircles an additional axial inlet of the preheated dilution air. A partially premixed combustion model (Zimont et al., [24]) available in ANSYS-CFX is used for this simulation. The grid consists of $3.6 \cdot 10^6$ tetrahedral elements, corresponding to $6 \cdot 10^5$ control volumes of the dual mesh.

As the current flow is essentially a free shear flow, without the need for inclusion of wall boundary layers, the ITS combustion chamber could also be computed using conventional LES models. However, industrial combustion chambers are significantly more complex, making a complete LES simulation including the inlet swirler, the piping and potential heat transfer predictions at the burner walls impractical.

The flow structures for the cold and the hot flow simulation at a given instance in time are shown in Figure 7. Experience shows that RANS models are not reliable in predicting the change in flow topology indicated by that figure. This can be seen in more detail in Figure 8, Figure 9 and Figure 10, showing the radial distributions of the statistically averaged velocity and temperature at a distance from the inlet, approximately equal to one ring diameter (note that the SST and the k- ε models are virtually identical for free shear flows). Superior accuracy of SAS results relative to the RANS simulation confirms that SAS is a viable method for such a complex flow. For another, significantly more detailed evaluation of the SST-SAS model for combustion chamber simulations, see Widenhorn et al. [23].

Figure 7: SAS solution for ITS combustion chamber, iso-surface Ω^2 -S²=10⁷ s⁻² : left - non-reacting, right - reacting.

Figure 8: Non-reacting flow velocity profiles at x=138 mm. Left – Axial velocity. Right – Tangential velocity

Figure 9: Reacting flow velocity profiles at x=103 mm. Left – axial velocity. Right – tangential velocity

Figure 10: Reacting flow temperature profile at x=103 mm.

Aerodynamic flows with massive separation

NACA0021 airfoil beyond stall

This flow was experimentally investigated by Swalwell et al. [22]. The symmetric NACA0021 airfoil was measured at a high angle of attack of α =60° at a Reynolds number of 2.7·10⁵. The spanwise extension of the computational domain was selected to be four chord-lengths for this calculation, and an O-type hexahedral grid 140×101×134, provided for the DESider consortium, was used for the SST-SAS simulation with the ANSYS-CFX solver. Freestream conditions were applied at the outer limit of the grid and periodic conditions in spanwise direction. A timestep equal to 3% of the convective timescale (chord length over the inlet velocity magnitude) was used.

Figure 11 shows a comparison of the computed and the experimental pressure distributions. The agreement is good and within the range of other simulations in the DESider project. Figure 12 shows the turbulent structures computed by the SST-SAS model behind the airfoil. The structures are essentially resolved down to the grid limit, with the larger structures indicating the grid coarsening away from the airfoil. Figure 13 shows the power spectral densities of the lift and drag coefficients, which are in good agreement with the data, demonstrating the correct temporal response of the model.

The experience gained during the simulation of this flow showed the importance of sufficiently long physical time integration for the correct prediction of the average surface pressure and of the low-frequency part of the spectra of forces. During the reported simulation, about 400 convective units have been run for the transient statistics after first establishing the solution. In order to achieve better averaging, the spectra of forces have been calculated for each grid section separately and then averaged along the spanwise direction.

The integral lift and drag coefficients, presented in Table 1, are predicted with 2% accuracy compared to the measurements. This good agreement in lift and drag might be partially coincidental, as other project partners have found a dependency

on the spanwise extent of the domain, which was not varied in the current simulations.

The good agreement of the power spectral densities is of major relevance for the validation of the SST-SAS model, as it demonstrates the accuracy of the model in the time/frequency domain.

Table 1 Lift and drag coefficients for the NACA0021 at 60° angle of attack

	Lift coefficient, C _L	Drag coefficient, C _D
SST-SAS	0.915	1.484
Experiment	0.931	1.517

Figure 11: Mean surface pressure coefficient

Figure 12: SAS-generated turbulent structures behind the airfoil

Figure 13: Turbulent spectra of forces for NACA0021 airfoil: left – power spectral density of the lift coefficient, right – power spectral density of the drag coefficient. The Strouhal number St is calculated using the free stream velocity magnitude and the chord length.

Full aircraft FA-5 configuration

A delta-canard (FA-5) configuration, which was investigated by Laschka et al. [12], constitutes a complex aerodynamic application simulated with the SST-SAS model. The configuration is shown in Figure 14. The airplane model has an angle of attack of α =15° and a Reynolds number based on the model length is Re=2.78·10⁶. A fine time step value used in this simulation corresponds to about 500 timesteps per convective time unit (the model length over the inlet velocity magnitude). The flow is of low Mach number and computed incompressible. The unstructured hybrid grid provided by EADS for a half-domain with a symmetry plane had 36·10⁶ elements (11·10⁶ control volumes). The application of a symmetry condition is warranted in this case, as the resolved turbulence is not adjacent to the symmetry plane.

Vortex structures, computed with the SST-URANS and the SST-SAS model, are shown in Figure 14. It can be seen that the SAS model resolves the turbulent structures in the vortex separated region. Figure 15 shows distributions of the axial velocity (with respect to the airplane) in comparison between the experiments and the SST-SAS simulation performed using ANSYS-CFX. The agreement between the simulations and the data is quite good considering the complexity of the applications and other simulations carried out within the DESider project. Still, the computations indicate a somewhat early break-up of the main vortex, relative to the experiments. This testcase will be further evaluated during the EU project ATAAC. It is planned to evaluate the influence of further grid refinement on the solution.

Figure 14: Resolved vortex flow configuration: left – SST-URANS, right – SST-SAS (note 12.5 times smaller maximal eddy viscosity in the right figure).

Figure 15: Mean axial velocity distributions in cut planes: left – experiment, right – SST-SAS simulation

Aeroacoustic applications

3-D acoustic cavity

Air flow past a 3-D rectangular shallow cavity is calculated in this test, with the cavity geometry and flow conditions corresponding to the M219 experimental test case of Henshaw [7]. The geometry dimensions of the M219 cavity are L×W×D = $5\times1\times1$ (length, width, and depth), with a depth D of 4 inches. The side boundaries are treated as symmetry planes, the top boundary is a far-field boundary, and all the solid surfaces as adiabatic non-slip walls. The amount of ambient space, included into the computational domain, is: 31'' from the inlet to the cavity leading edge, 21'' from the cavity trailing edge to the outlet, 68'' from the cavity opening level to the top boundary. Significant space equal to 16'' is left between each side boundary and the correspondent cavity edge to prevent any influence of the symmetry conditions used at the side boundaries. The inlet Mach number is 0.85, and the Reynolds number per one meter is $13.47 \cdot 10^6$.

The grid consists of $5.8 \cdot 10^6$ hexahedral elements. The time step for the simulation is $2 \cdot 10^{-5}$ s, which is 18 times less than the hydrodynamic time scale based on the inlet velocity and the cavity depth. Ten thousand time steps have been calculated to obtain a developed flow state, and another ten thousand steps afterwards – to obtain statistics for the spectral analysis of a pressure field.

Figure 16: Resolved turbulent structures: iso-surface Ω^2 -S²=5·10⁵ s⁻².

Figure 17: Power spectral density of the transient wall pressure signals on the cavity bottom: left – sensor K20 located close the front wall, right – sensor K29 located close to the rear wall.

Figure 16 shows the turbulent structures, produced by the SST-SAS model. The power spectral density of the transient pressure signals calculated and measured at the K20 and K29 sensor locations on the cavity bottom near the leading and the downstream wall respectively, is plotted in Figure 17. These plots show that the PSD levels are captured in good agreement with the data. However, the main acoustic modes are shifted relative to the experiments by ~10%. The reason for this shift is not entirely clear and will also be subject to further studies in the ATAAC project. The results, presented in Figure 16 and Figure 17, have been calculated using ANSYS-FLUENT. A similar SST-SAS simulation with ANSYS-CFX, carried out and published earlier (Egorov and Menter, [2]), delivered the comparable accuracy.

Generic car mirror

In this test case the SST-SAS model of ANSYS-CFX was used to simulate the transient behaviour of the flow around a generic car side mirror. One known experiment of this type was conducted at Daimler-Chrysler Aerospace and published in (Höld et al., [8]) and (Siegert et al., [20]). The studied case is a half cylinder with a diameter *D* and length of 0.2 m, blunted by a quarter of a sphere with the same diameter. This generic body, having roughly the shape of an external car mirror, is mounted on a flat plate with 1.6m width and 2.4m length. The rear-face foot of the cylinder is located 0.9 m from the leading edge of the plate. The mirror is exposed to a free-stream air velocity U_0 =140 km/h at zero incidence, leading to the Mach number and the Reynolds number of 0.11 and 5.05 $\cdot 10^5$ respectively. The flow geometry, the computational domain, and the multi-block hexahedral grid of $3 \cdot 10^6$ elements are illustrated in Figure 18 and Figure 19.

The predicted mean surface pressure distribution, as well as the noise spectra at several monitor points in the near field are available from the experiment. The locations of the selected transient pressure sensors and the mean pressure probes are shown in Figure 20 and Figure 21 respectively.

The chosen simulation timestep of $\Delta t=0.02 \cdot 10^{-3}$ s was found overly conservative, and was mainly used for the sake of the direct comparison with simulation results obtained by other authors. The Courant number in the resolved turbulence zone was around 0.2, and one test run with the five times higher timestep of $0.1 \cdot 10^{-3}$ s delivered nearly the same resolution quality. As indicated by Grahs and Othmer [5], the practically relevant noise frequency for this flow type goes up to 4 kHz. The increased timestep, corresponding to a Courant number of one, is only 2.5 smaller than the highest frequency noise period. This indicates a need to further refine the grid locally in the separation zone to also resolve the highest acoustic modes. The surface pressure spectra are therefore plotted here in Figure 24 only up to the grid-relevant limit of 1 kHz. The overall simulation time was 0.8 s (156 convective units). The pressure signals were recorded for spectral analysis during the last 0.3 s (58.5 convective units).

The mean pressure distribution over the mirror surface agrees very well with the measured values, as shown in Figure 22. Unsteady resolved turbulent structures are shown in Figure 23. Pressure spectra for sensor locations 119, 121 and 123 (see Figure 20) are presented in the form of sound pressure levels in Figure 24. Multiple Fourier spectra were extracted for separate time sequences of 4096 timesteps using the Hanning window function and then averaged to filter out the spurious noise. The resulting SPL amplitudes agree with the experiment similarly as the published DES results by other authors (see, for instant (Rung et al., [16]) and (Ask and Davidson, [1]).

Figure 18: Schematic of the experiment

Figure 19: Grid details near the mirror

Figure 20: Location of some of the transient pressure sensors on the plate surface. Pressure spectra are presented in Figures 6-8 for the marked sensors 119, 121 and 123.

Figure 21: Location of the mean pressure sensors: left - front face, right - rear face.

Figure 22: Mean pressure distribution

Figure 23: Turbulent structures behind the mirror

Figure 24: Transient pressure spectrum at different sensor locations (indicated above sub-pictures). Lower right:.

Summary

Several testcases computed with the SAS model formulation have been presented. They demonstrate the models ability to resolve turbulent structures in unstable regions of complex engineering flows. The testcases typically show good agreement in the mean values and spectral quantities as compared to experimental data. The SAS model offers an attractive alternative to existing "hybrid" RANS/LES methods for strongly unstable flows, as it offers a RANS fallback solution independent of the grid spacing. It is therefore a valuable engineering tool allowing engineers a relatively safe passage into scale-resolving simulations of technical flows.

Acknowledgment

The current work was partially supported by the EU within the research project DESider (Detached Eddy Simulation for Industrial Aerodynamics) under contract No. AST3-CT-200-502842 (http://cfd.mace.manchester.ac.uk/desider/)

Cooperation with our partners and colleagues is highly appreciated, in particular for the NACA0021 grid provided by NTS S.-Petersburg, the airplane FA-5 grid by EADS Deutschland (Military Airplanes), the AITEB turbine blade grid by MTU Aero Engines, the grid for a buoyant jet flow by ETH Zurich. The swirl burner, the car mirror and the acoustic cavity simulations were prepared by our colleagues at ANSYS R. Bender, T. Belamri and G. Link.

References

- 1. Ask, J. and Davidson, L.: The sub-critical flow past a generic side mirror and its impact on sound generation and propagation, AIAA Paper 2006-2558 (2006).
- Egorov, Y. and Menter, F.: Development and application of SST-SAS turbulence model in the DESIDER project, in "Advances in hybrid RANS-LES modelling", Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Vol. 97, pp. 261-270, Springer, (2008).
- Fletcher, D. F. and Lasuye, T. and Torré, J-P.and Xuereb, C.: CFD modelling of partially baffled stirred vessels using the Scale Adaptive Stimulation (SAS) turbulence model. In: 11ème Congrès de la Société Française de Génie des Procédés, 9-11 Oct 2007, Saint-Etienne, (2007),.
- Frank, T., Lifante, C., Adlakha, M., Prasser, H.-M. and Menter, F.R.: Simulation of Turbulent and Thermal Mixing in T-Junctions Using URANS and Scale-Resolving Turbulence Models in ANSYS CFX" - Experiments and CFD Codes Application to Nuclear Reactor Safety OECD/NEA & International Atomic Agency (IAEA) Workshop Sept. 2008, Grenoble, France, pp. 23, (2008).
- Grahs, T. and Othmer, C.: Computation of acoustic sound sources: Parametric study concerning the aero acoustic properties of wing mirror geometries, Volkswagen AG, Wolfsburg, VDI Berichte, Vol. 1967; part 1, pp. 23-36, ,06).
- Haase, W., Braza, M., Revell, A. (Eds.): DESider A European effort on hybrid RANS-LES modelling, Results of the European-Union funded project, 2004-2007, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Vol. 103, Springer, (2009).
- Henshaw, M. J. de C. : M219 cavity case, In: "Verification and validation data for computational unsteady aerodynamics", Tech. Rep. RTO-TR-26, AC/323/(AVT) TP/19, pp. 473-480, (2000).
- Höld, R., Brenneis, A., Eberle, A., Schwarz, V., Siegert, R.,: Numerical simulation of aeroacoustic sound generated by generic bodies placed on a plate: Part I – Prediction of aeroacoustic sources, AIAA Paper 99-1896, (1999).

- Honkanen, M., Heikkinen, J., Saarenrinne, P. and Korpijärvi J.:Time-resolved stereoscopic PIV experiments for validating transient CFD simulations, 7th International Symposium on Particle Image Velocimetry Roma, Italy, September 11-14, (2007).
- Imberdis, O., Hartmann ,M., Bensler, H., Kapitza L. and Thevenin D. A Numerical and experimental investigation of a DISI-engine intake port generated turbulent flow, SAE Paper 2007-01-404, Warrendale, Pa. (2007).
- Joo, J. and Durbin, P.: Simulation of Turbine Blade Trailing Edge Cooling, J. Fluids Eng., Volume 131, Issue 2, (2009).
- Laschka, B., Ranke and H., Breitsamter, C., : Application of unsteady measurement techniques to vortical and separated flows, Z. Flugwiss. Weltraumforsch. 19, 90-108. (1995).
- Magnoli, M.V. and Schilling, R.: Vortex shedding in a Francis Runners Trailing Edges, IAHR 24th Symposium on Hyd. Machinery and Ssystems, Foz do Iguassu, (2008).
- Martini, P., Schultz, A., Whitney and C.F. and Lutum, E.: Experimental and Numerical Investigation of Trailing Edge Film Cooling Downstream of a Slot with Internal Rib Arrays, 5th European Turbomachinery Conf., Prague, 18th-21st March 2003, (2003).
- Menter, F.R. and Egorov ,Y.: The scale-adaptive simulation method for unsteady turbulent flow predictions. Part1: Theory and model description, Journal of Flow Turbulence and Combustion current issue, (2009).
- Rung, T., Eschricht, D., Yan, J. and Thiele, F.: Sound radiation of the vortex flow past a generic side mirror, AIAA Paper 2002-2549, (2002).
- Rusch, D.:Turbulence model validation for fire simulation by CFD and experimental investigation of a hot jet in crossflow, Ph.D. Theses, Diss. ETH No. 16966, Zurich, (2007).
- Schildmacher, K.-U., Koch, R., Wittig, S., Krebs, W. and Hoffmann, S., Experimental investigations of the temporal air-fuel mixing fluctuations and cold flow instabilities of a premixing gas turbine burner, ASME Paper 2000-GT-0084, (2000).
- Schildmacher, K.-U.: Experimentelle Charakterisierung der Instabilitäten vorgemischter Flammen in Gasturbinen-Brennkammern, Forschungsberichte aus dem Institut für Thermische Strömungsmaschinen, Band 26/2005 (Ph.D. Theses, in German), (2005).
- Siegert, R, Schwarz and V. Reichenberger, J.: Numerical simulation of aeroacoustic sound generated by generic bodies placed on a plate: Part II – Prediction of radiated sound pressure, AIAA Paper 99-1895 (1999).
- Smirnov, P. E., Kapetanovic, S., Braaten, M. E., Egorov, Y., Menter, F. R., Application of the SAS turbulence model to buoyancy driven cavity flows, ASME Paper GT2009-59621 (2009).
- 22. Swalwell, K. E., Sheridan, J. and Melbourne, W. H.: Frequency analysis of surface pressures on an airfoil after stall, AIAA Paper 2003-3416, (2003).
- 23. Widenhorn, A., Noll, B, and Aigner, M.: Numerical Study of a Non-Reacting Turbulent Flow in a Turbine Model Comustor, AIAA Paper 2009-647, Orlando Florida, (2009).

 Zimont, V. L., Biagioli, F. and Syed, K.: Modelling turbulent premixed combustion in the inter-mediate steady propagation regime, Progress in Computational Fluid Dynamics 1: 14-28, (2001).