Extended Abstract for 43^{rd} AIAA Fluid Dynamics Conference, Topic Wing Aerodynamics

Simulation of Wing Stall

Rolf Radespiel, Daniela Gisele François, David Hahn, Peter Scholz
Institute of Fluid Mechanics, Technische Universität Braunschweig, 38108 Braunschweig, Germany

Katharina Wawrzinek, Thorsten Lutz
Institute of Aerodynamics and Gasdynamics, Universität Stuttgart, Stuttgart, Germany

Torsten Auerswald, Jens Bange
Institute of Geoscience, Universtät Tübingen, Germany

Christoph Knigge, Siegfried Raasch,
Institute of Meteorology and Climatology, Leibniz Universität Hannover, Hannover, Germany

Philip Kelleners, Ralf Heinrich
Institute of Aerodynamics and Flow Technology, DLR, 38108 Braunschweig, Germany

Silvia Reuss, Axel Probst, Tobias Knopp
Institute of Aerodynamics and Flow Technology, DLR, 37073 Göttinge, Germany

Simulation capabilities for low-speed aircraft stall prediction are important for determining the limits of safe aircraft operations during design processes. The simulations are extremely demanding in terms of physical models involved, overall computation effort, and the needed efforts for validation. The present paper describes coordinated, fundamental research into new simulation methodologies for wing stall that also include the effects of atmospheric gusts. The research is carried out by the DFG funded Research Unit FOR 1066 composed of German Universities and the German Aerospace Centre, DLR. The research Unit investigates advanced models of turbulence, advanced physics-based gust models, and new numerical approaches for gust simulation. These modeling and computational activities are supplemented by an unique validation experiment, that aims at providing stall data on a high-lift wing with well defined, generic distortions of the onset flow.

I. Introduction

THe prediction of stall behaviors at low speed is an important part in the design of commercial transport aircraft. Take-off and landing performance is generally a strong driver of aerodynamic design. The ability to predict maximum lift and the associated angle of attack along with all force and moment coefficients within aerodynamic design cycles can accelerate industrial design processes and avoid costly design changes later on.

Aircraft design regulations require careful verification of manoevrability and the assessment of aircraft loads over the complete operational range. This involves the quantification of gust effects within the aircraft design process. Moreover, current practice for defining the operational margins to stalling speeds at climb and approach are also based on the current knowledge of gusts encountered during these flight phases. The research hypothesis followed with the present work is that improved characterization of gusts and improved capabilities in the simulation interactions between maximum lift and gust will eventually improve flight safety and allow for better exploitations of aircraft take-off and landing performances.

The German DFG Research Unit 1066 [1] addresses two fundamental research areas of aerodynamic stall effects. These are the simulation of aircraft wing stall and the simulation of aircraft nacelle stall including the assessment of engine fan stability. The first research area is reviewed within the present paper while the second one is covered in a separate companion paper [2].

The research approach of DFG Research Unit 1066 requires a coordinated effort in the area of physical modeling of turbulent flows, in advanced numerical simulation systems and in experimental validation. The area of physical flow modeling not only covers the high Reynolds number flows of aircraft wings but also the modeling of atmospheric disturbances. Hence, four university institutes and the German Aerospace Centre, DLR, contribute to the work presented here. Technische Universität Braunschweig contributes an advanced Reynolds stress model (RSM) of turbulence. As wing stall usually involves complex 3D boundary layers, juncture flows and the effects of longitudinal vortices it is assumed that only RSM will eventually result in satisfactory capture of all these flow features. Current research on the RSM addresses the question whether hybridization of RSM and local DES can improve the prediction of strong flow separations. Advanced models for atmospheric distortions are provided by Leibniz Universität Hannover and Universität Tübingen. Here, two approaches are followed: Universität Tübingen contributes a statistical model of atmospheric turbulence that is based on flight data of a sophisticated probe located well below a helicopter in low level flight. Leibniz Universität Hannover, on the other hand, uses high resolution Large Eddy Simulations of the convective atmospheric boundary layer to extract and characterize gust samples and complete 3D atmospheric flow fields. These atmospheric disturbance models are then input to aerodynamic wing simulations with the DLR-code TAU. With the TAU code, atmospheric disturbances can be simulated by using moving Chimera grids. These Cartesian grids carry the disturbances from the wing farfield to the wing proximity. The Chimera approach allows to take all aerodynamic interactions of atmospheric disturbances and the wing flow field into account. Additional research work followed by DLR implements and investigates a range of gust interaction models that aim for improved computational efficiency.

The complex simulation approaches of the present work ask for validation, in particular as we address atmospheric free stream distortions where the smallest resolved length scale is in the order of the wing chord. Therefore, Technische Universität Braunschweig set up a wind tunnel experiment by which controlled disturbances are generated upstream of a high-lift airfoil. This set up is an experimental representation of a generic 2D gust. Two groups, Universität Stuttgart and the DLR in Göttingen use this experimental set up to perform numerical rebuilding of the wind tunnel tests with the generic gust representation.

II. Turbulence Modeling for Stall Simulations

A. RANS model

The aerodynamics of wings at stall are usually characterized by a broad range of flow scales, which are difficult to capture in numerical flow simulations. While the unsteady behaviors of separating, turbulent boundary layers go along with vortex shedding and flow hysteresis a consistent turbulence model for these flows is still not yet established. Unfortunatetely, for aircraft flows with Reynolds numbers around 10^7 and 10^8 there is no real alternative to using RANS models of turbulence for the global flow field. We note that wing flows at high lift are generally much affected by vertical flow in junctures and longitudinal vortices that stem from flap edges, nacelle strakes or from other desired or unavoideable geometric discontinuities. Moreover, wing stall is usually affected by nonisotropy of normal stresses, nonalignments of stress and strain and nonequilibrium of turbulence production and dissipation. It has become now more and more common understanding that only Reynolds stress models of turbulence (RSM) can represent these effects in a suited way to predict the onset of flow separation with sufficient accuracy. In the work of the DFG Research Unit 1066 we use the JHh-v2 RSM [3] which is an extension of the Jakirlic-Hanjalic-homogeneous RSM [4]. The model applies a length-scale-supplying equation for the homogeneous dissipation rate ε^h and low-Reynolds damping in order to accurately model near-wall turbulence. Transition of laminar to turbulent boundary layers is formulated such that turbulent stresses are initiated in a physically sound way at the transition location and the resulting numerical flow solution is unique [5]. This RANS model was scrutinized for a broad range of flows: Subsonic diffuser, single subsonic and transonic airfoils, multi-element airfoils, transonic swept wings, nacelle of transport aircraft and a solid data base of model performance for conventional validation cases does now exist.

B. Hybrid RAN/LES model

It is now widely accepted that local Large Eddy Simulations present an efficient way to resolve the dynamic behavior of large separated flow areas. To apply the JHh-v2 RSM in a local detached eddy simulation (DES), the (homogeneous) dissipation-rate is multiplied by the ratio of the RANS length scale to the hybrid length scale of the respective DES-model. The hybrid length scale is here provided by the algebraic delayed DES approach (ADDES) [7]. It uses a DES definition given by a combination of algebraic boundary-layer criteria, which are evaluated along wall-normal lines. This approach succeeds in detecting pressure induced separations at smooth aerodynamic surfaces, that are usually not recognized by the common DES sensors. Thus, the RANS modeling can be explicitly used in attached regions and the interface to LES is placed at the separation point.

A major problem of current research is the occurrence of the so called “grey” transition zone from RANS to local LES regions. In this transition zone, the modeled Reynolds stresses are suppressed by the hybrid length scale formulation but the resolved turbulent fluctuations have not yet grown to their physical levels. A crucial ingredient of a useful hybrid scheme is hence to force the generation of turbulent content in the LES branch, within the separated flow region. At present, two models are under investigation: In the stochastic SGS model the Reynolds stresses are multiplied with independent, identically distributed random numbers with an expectation value of 1 recomputed at each node and in each physical time step. The second, alternative model is to provide synthetic turbulence at the inflow to the local LES region. The strategy is to extract stress levels and turbulent length scales from the close-by RANS region and use that information to inject physics-based fluctuations into the LES region. Both approaches are being followed and compared within the current research. Figure 1 displays some sample results of this work.

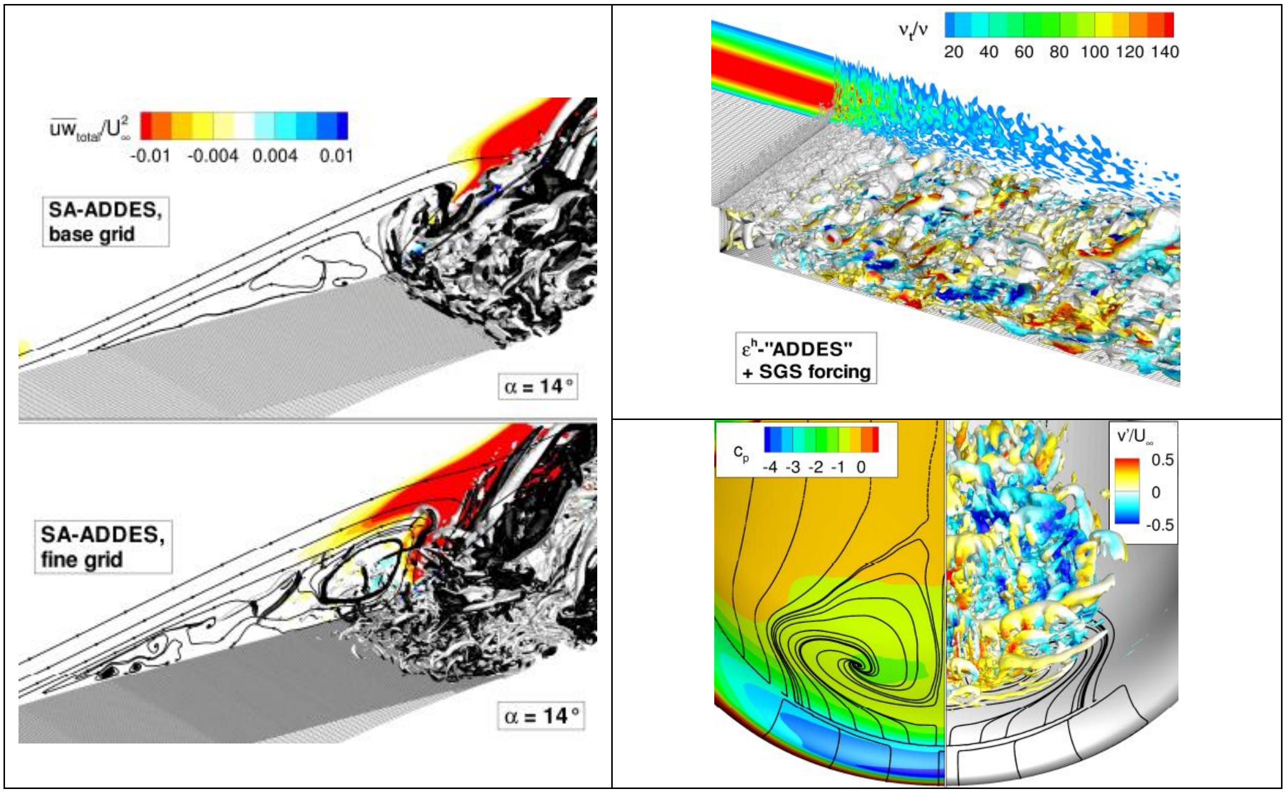


Figure 1: Selected hybrid RSM-LES solutions: Separated flow over a backward facing step (upper right), separation onset on the single airfoil HGR01 (left) and flow separation inside a stalling nacelle inlet (lower right).

The full paper will contain detailed descriptions of the turbulence models along with more details of the test cases to assess the problem of stress depletion in the grey regions of the hybrid RSM-LES models.

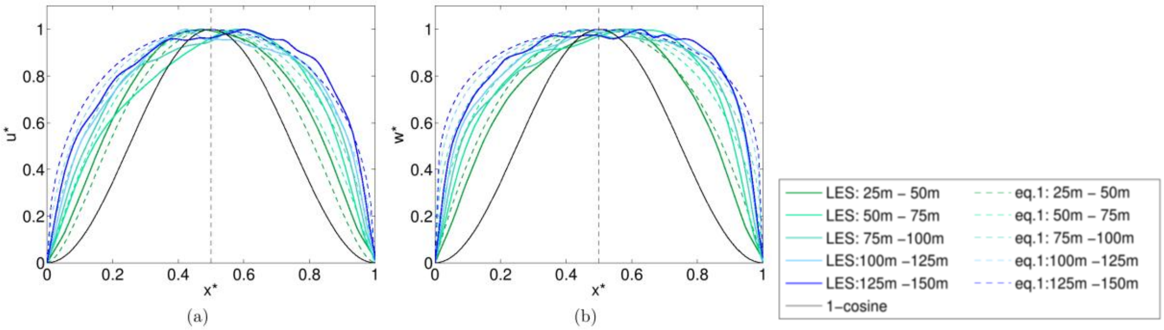
III. Gust Modeling

Modern numerical simulation methods and advanced inflight measurements provide the opportunity to obtain a physics based characterization of the atmospheric boundary layer with its strong, discrete wind pulses also known as gusts. The well-known (1-cosine) law which is described in the Federal Aviation Regulations for Transport can thus be scrutinized and improved if necessary. Furthermore, LES data of the atmospheric boundary layer enable analysis of the two dimensional structure of the of the wind gusts. This is an improvement over the previous approach where measurements only provided one dimensional data.

The atmospheric LES solver PALM used for the present work has been developed at the Institut für Meteorologie und Klimatologie at the Leibniz Universität Hannover [8], [9]. PALM solves the filtered non-hydrostatic, incompressible Boussinesq equations, the first law of the thermodynamics and the equation for turbulent kinetic energy (TKE). It scales very well on parallel computers and hence it can be used for simulations with a resolution in the range of 1-2 meters.

A relevant meteorological scenario causing intensive turbulence is a stormy low-pressure system. In the lowest part of the atmosphere, strong wind shear caused by the high magnitude of the wind and the friction at the bottom, leads to intensive turbulence. This purely dynamical driven meteorological scenario is caused by a strong geostrophic wind which is set to 30 m/s in the simulation setup. Neutral boundary layer conditions up to 700 m are followed by an inversion of 2 K/100 m in the initial temperature profile.

To obtain the information about discrete gusts from the time depending three dimensional data output of the simulation, horizontal cross sections at different heights are extracted. Several one dimensional virtual measurements in these instantaneous flow fields provide space depended information of the respective velocity component. With this method, the application of the Taylor hypothesis is not required. The discrete wind gusts contained in these one dimensional evaluations are defined following the approach of Camp 1968 [10]. Five different gust classes corresponding to lengths between 25 m and 125m have been statistically defined. These are displayed in Figure 2. Beside the gust shapes calculated from the LES data, an analytical model of the gusts is also shown along with the classical 1-cosine law.



(a) (b)

Figure 2: Nondimensional velocity profiles of the streamwise and vertical velocity profiles in gusts with different size at 30m altitude in the stormy atmosphere.

These results show that the one-dimensional mean gusts shapes are more complex than the simple 1-cosine law suggests. The possibility to analyze the two dimensional gust shape is also used. First results are given in Figure 3. They exhibit mean vertical gust shapes at an altitude of 30 m. The full paper will present the gust representations in more detail. The two-dimensional characterizations are important for future manoeverability assessment of transport aircraft.

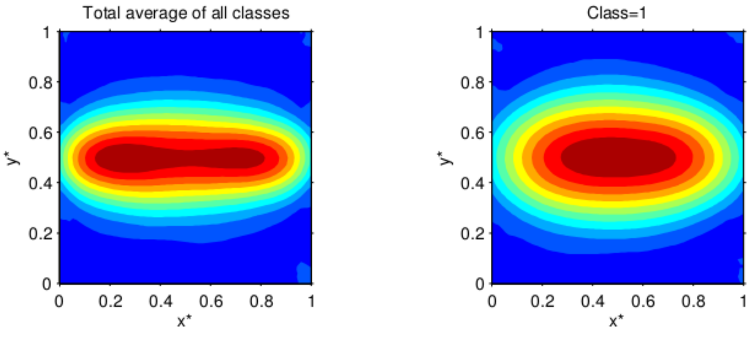


Figure 3: Shape of vertical velocity contours in 2D gust analysis at 30m altitude in the stormy atmosphere.

An alternate approach is based on inflight measurements for atmospheric flow field characterization. Data are measured by the Helipod [11]. This is an airborne measurement probe which is attached to a helicopter. Measurements are taken at altitudes between 40 m and 2000 m. Several sensors (e.g. wind vector, temperature, humidity) sampled at 100 Hz are mounted for high-resolution measurements in the ABL. A large data set of measurements for different meteorological scenarios is available including atmospheric flow in various thermal stratifications (stable, very stable, neutral, convective). As basis for the generation of the turbulent wind field from this inflight data base the Fourier approach is used, in which the sum is taken over several waves with different wave numbers to build a random velocity field. The resulting wind field is divergence-free and isotropic. By applying the Cholesky decomposition, the wind field can be modified to possess a predefined correlation matrix which introduces anisotropy. With a suited scaling of the resulting wind field the variances from the measurement can be met [12].

A cross validation of the synthetic turbulence field with PALM data is displayed in Figure 4. Here, a PALM simulation result was used to extract generic flight data along the black lines shown on the left of the figure. These data were then used to create the synthetic wind field displayed on the right. While the given statistics are reproduced very well, some features like coherent structures cannot be generated (see comparison synthetic turbulence/LES).

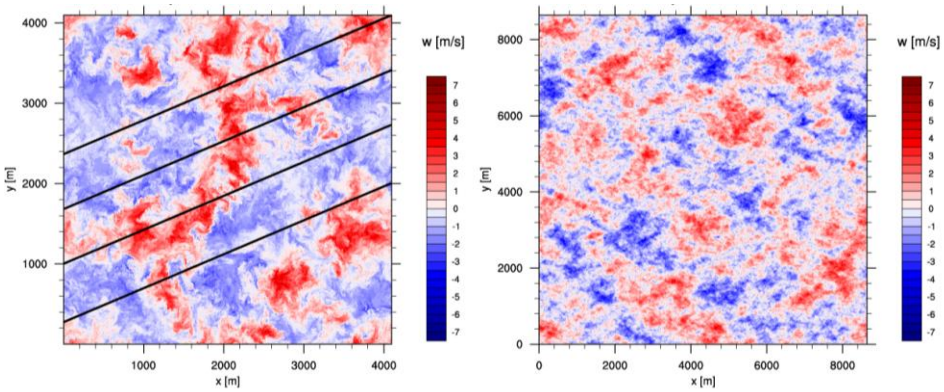


Figure 4: Contours of vertical gust velocities in a PALM LES (left) and for the corresponding synthetic wind field model (right) .

IV. Simulation Approaches for Wing Stall including Gusts Effects

The prediction of unsteady loads caused by atmospheric effects like gusts is essential for aircraft development. Gust loads are important for the design of the structure but also for the layout of the control surfaces and the Flight-Control-System (FCS). With respect to maximum loads, especially gusts of high amplitude and wave-length being long relative to the reference chord length are of interest. But also gust of shorter wave-length can be of relevance, if the aircraft is flying close to maximum lift. In that case even gusts of short wave-length can trigger wing-stall. In the DFG Research Unit 1066 methods for the prediction of the unsteady behavior for both situations (short and long wave-length) are developed, based on the DLR TAU-Code.

One method for modeling of gusts is the so called Disturbance Velocity Approach (DVA), see for example [13]. This is rather easy to implement in CFD codes and allows the usage of standard meshes for aerodynamic analysis. The method captures the influence of the gust on the aircraft but it does not model the feedback of the aerodynamics of the aircraft on the gust shape. Therefore, for gusts of short wavelength a certain prediction error can be expected. Hence an alternate approach was also implemented in TAU: The gust is be fed into the discretized flow field using an unsteady boundary condition at the farfield. The advantage of the method is that the mutual interaction of gust and aircraft is captured. However, a high resolution in the whole domain is required, in order to accurately convect the gust from the inflow boundary to the aircraft. It is planned to implement an alternative third approach for gust modeling in TAU, which also allows a mutual interaction of gust and aircraft, but at a lower computational cost, compared to the approach of numerically resolving the gust. This approach called “source gust model” was initially developed by Jirasek [14].

To compare these methods, a generic 2D test case representing a wing and horizontal tail plane (HTP) is used. The grid, as shown in Figure 5, is an overset mesh. Component meshes for wing and HTP are placed into a Cartesian background mesh. An additional grid (green) with a high resolution in flow direction is used for the “transport” of the gust from the farfield boundary to the wing-HTP configuration. An initial grid density study revealed that a resolution of 100 cells for one gust wave-length has to be used to avoid viscous losses for the resolved gust (RG) approach. Computations were made for 3 different gust wave-lengths (1, 2 and 4 wing chord-lengths). The classical (1-cosine) gust shape was selected. The gust amplitude is 10% of the on-flow velocity. Figure 5 shows the comparison of results of the DVA (dashed lines) and the more accurate RG approach, that makes use of the unsteady boundary condition. The resulting lift history for the three wave length is plotted versus dimensionless time. Surprisingly good agreement is found for both numerical gust representations, indicating a need to resolve gust for wavelengths in the order of one airfoil chord only.

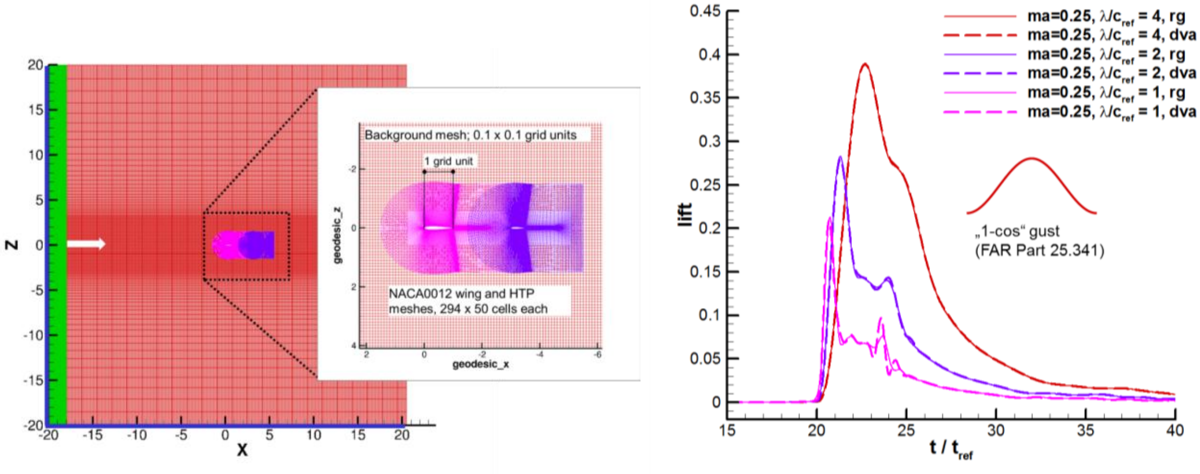


Figure 5: Computational grid and time response of a 2D configuration composed of two airfoils representing wing and HTP to 10% vertical gusts with different wave lengths, M_∞=0,25

A disadvantage of the RG approach described above is that for gusts or other atmospheric effects of short wave length a very high grid resolution is required. As TAU code has a convergence order of two it appears that freely convecting vortices suffer from rapid decay on grids of practical mesh density. Increasing the mesh density to obtain an accurate representation of these flow structures leads to excessive computational cost, particularly in three dimensions. Higher-order methods, can much better represent these structures, and thus offer the potential to lower the computational cost. To combine the strengths of second order and higher order methods a zonal approach is pursued where the flow domain of interest is divided and in each region the most suited discretization method is applied. In the implementation of this zonal approach the boundaries of the different grids overlap, and the developing flow state is coupled in these overlapping regions, as in Figure 5. Details of the domain coupling procedure are described in Ref. [15]. The recently developed higher-order Cartesian grid module of TAU, called CTAU uses compact finite-difference Padé-type schemes in conservative finite-volume formulation [16]. Due to the nonlinear convective fluxes the resulting Cartesian grid method is limited to being fourth order accurate. Present experience with CTAU indicates drastic improvement in resolution and conservation of fine flow structures, including gust shapes and amplitudes, as these are convected over large distances. In the full paper results of using this higher order method will be compared to the methods described above for gust interaction. It is planned to apply this method for wing stall cases triggered by gusts of short wave length.

V. Validation Experiments

A new validation experiment is set up within the Research Unit 1066 that aims at studying the effects of short-wave-length gusts on high-lift airfoil stall. The gust is represented by a transversal vortex that moves with the onset flow to the research airfoil. Creating transversal quasi two-dimensional vortices in a closed wind tunnel test-section is a task, that needs considerable effort. Figure 6 displays the principle of the test setup used for the present research, which is composed of a vortex generator [17] and a high-lift airfoil. A symmetric two-dimensional and rotatable NACA 0021 airfoil (vortex generator airfoil) with a chord length of 0.3m is used to create quasi two-dimensional transversal vortices. As the vortex generator airfoil changes its angle of attack and its circulation,a layer of vorticity is created in the vortex generator wake. Crucial are here the pitching rate, which determines the spatial extent of the created disturbance and the pitching amplitude, which determines the strength of the wake vortex. Our current set up obtains 10° pitch amplitude at a rate of 0.7°/ms which creates a wake structure with a length in the order of the high-lift airfoil chord length.

This motion performance is obtained by using a light-weight vortex generator airfoil along with digital linear actuation. Four linear servo actuators of the type Copley XTA 3806 are arranged in pairs, one pair to the right and one to the left side of the vortex generator airfoil as shown in Figure 7. For operating several actuators in parallel, synchronization is provided by a master-slave control system. Failure modes are taken into account by digital and mechanical means to prevent too large or asynchronous actuation motions or excessive mechanical loads on the airfoil. Details are given in Ref. [17].

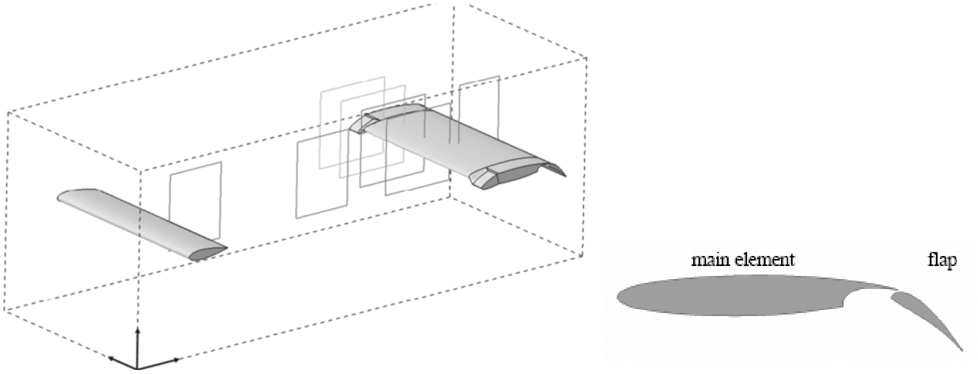


Figure 6: Geometrical sketch of experimental set up to generate vortical disturbances in the airfoil flow around a high-lift airfoil and of the used PIV planes.

The high-lift airfoil has the DLR F15 geometry. It is equipped with 85 static pressure tabs and 8 time resolving Kulite XCW 93 transducers which are located on the main element and on the high-lift flap [18]. Figure 4 also displays the plane where PIV is used to measure phase-resolved flow fields. One test campaign was performed without the high-lift airfoil, to measure the disturbances in the empty test section. Two test campaigns yielded the interaction of vertical disturbances. Figure 8 displays two sample results obtained during this campaign, these are time histories of phase-averaged pressure data at the main wing nose and phase-averaged velocity data [18].

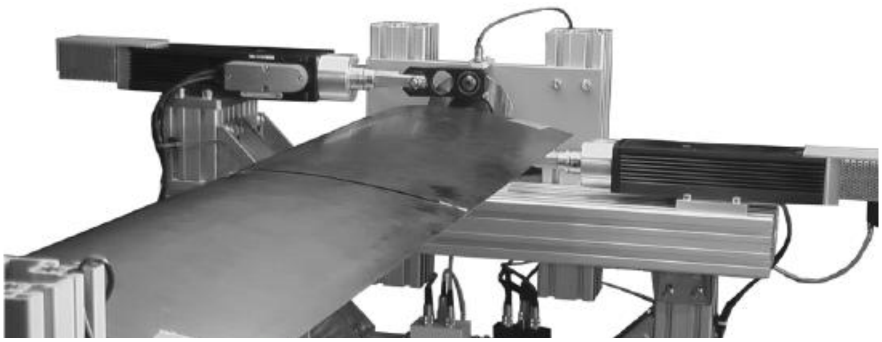


Figure 7: Motion apparatus used for the vortex generator airfoil

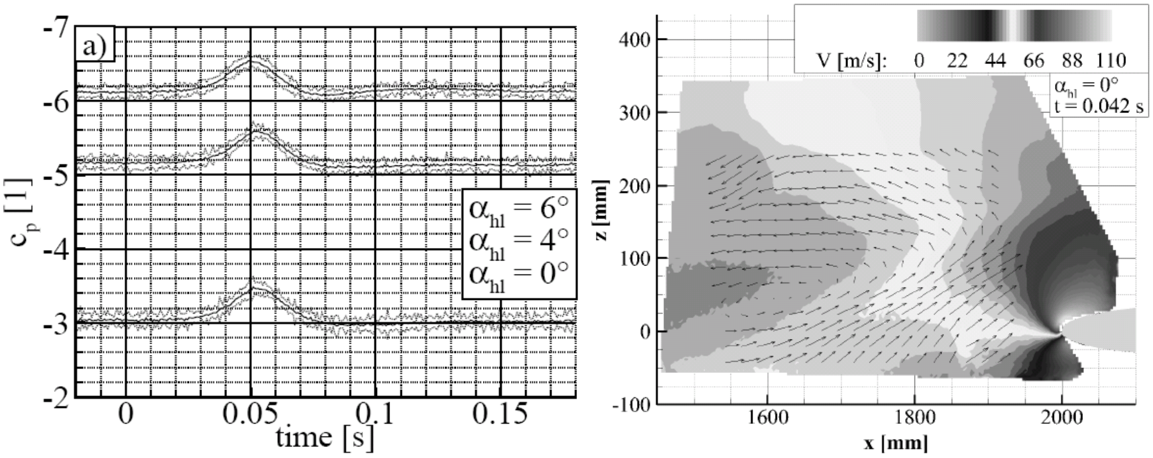


Figure 8: Sample experimental results for phase-resolved pressure at the main airfoil nose (left) and velocity field at nose (right).

VI. Validation Results

The DFG Research Unit 1066 aims at the validation of the advanced computation models for high-lift flows, where stall onset is affected by distortions of the freestream. Such a flow is experimentally investigated as described in Section V.

In the first validation step the steady wind tunnel flow with a high lift airfoil is considered. Note, that in the experiments, a special device was used to minimize the effect of the wind-tunnel side walls on the flow in the centerline section of the F15 wing. This device is a local droop nose that extends from the wall over 10% of the total wing span of the model. The droop nose is used to remove the nose suction peak in the vicinity of the wind tunnel wall, that is known to cause local separation of the juncture flow. In Figure 9 the flow pattern on the surface of the F15 airfoil predicted by the 3D JHh-v2 RSM simulation is compared with oil flow visualizations [19]. The development of secondary flow along the model-wall junction is in good agreement with the experimental result. This is not the case for a corresponding Menter-SST computation which massively over-predicts the juncture flow separation (not shown here). The pressure distribution from this 3D simulation is compared to a steady 2D simulation of the centerline section in Fig. 10. There is only a small offset between the 2d and 3D simulations indicating that the 3D effect of the sidewall boundary layer is indeed rather small. Both simulations deviate from the experimental pressure distribution close to the flap trailing edge. The reason for this discrepancy is still under investigation.

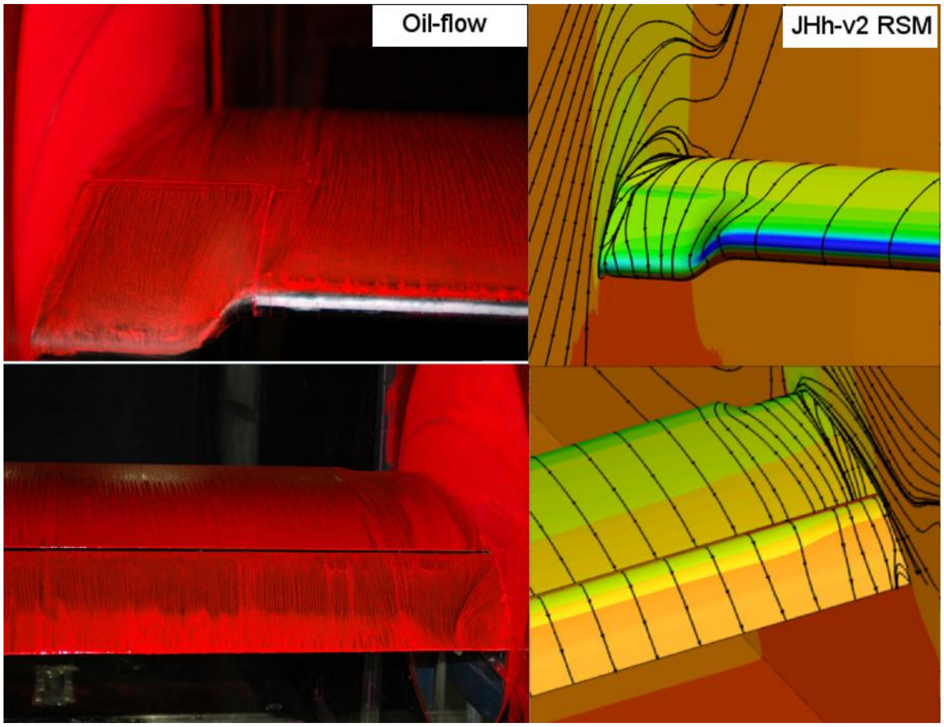


Figure 9: Comparison of oil-flow pictures and skin-friction lines from a 3D JHh-v2 RSM simulation at α=6°.

For a numerical rebuilding of the unsteady flow the Chimera approach is again used. The flow is computed on overlapping grid blocks and interpolation is used to exchange the solution between the blocks. This approach allows flexible meshing for resolving the various flow phenomena involved in the experiment. Figure11 displays the set up used to capture the flow over the vortex generator airfoil, the unsteady wake flow and the high-lift airfoil. The composite Chimera grid shown consists of a background wind tunnel grid with significantly finer grid blocks for the gust generator profile (yellow) and the two-element airfoil (red). Additionally, one well-resolved grid block (blue) is used to capture the convection of the free vortex. By moving that local grid with the flow the strength of the vortex sheet is accurately preserved during convection downstream. Note that in their overlappping regions, the three grids have similar element sizes to retain a consistent numerical resolution. Figure11 shows a snapshot of the interaction of the vortex with the high-lift airfoil. This is visualized by contours of the vorticity, along with the streamwise velocity component. While the unsteady flow computations shown in Figure 11 have employed the eddy-viscosity type SST model of Menter, computations using the JHh-v2 RSM will be also presented in the final paper and compared to the experimental data. Further results will address to the effects of the vertical vortex generator position relative to the DLR F15 airfoil. The computations will also cover the case where the unsteady vortex sheet hits the airfoil directly.

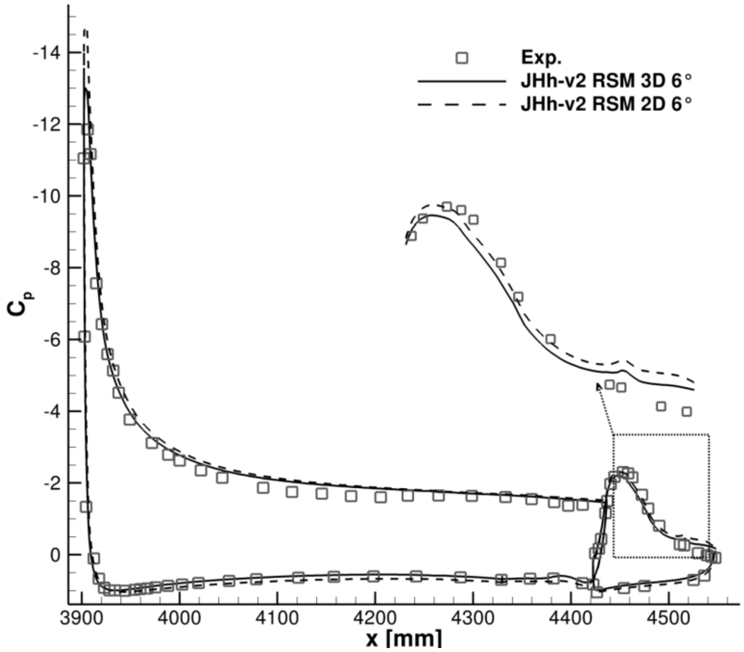


Figure 10: Comparison of pressure distributions from 2D and 3D simulations of the DLR-F15 airfoil using the JHh-v2 RSM.

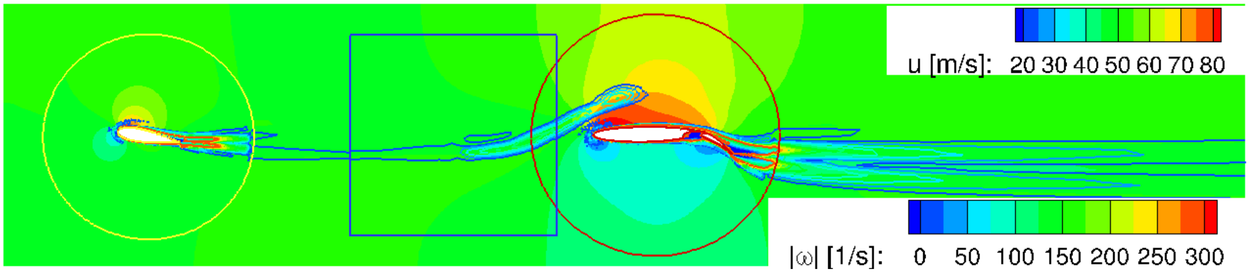


Figure 11: Chimera simulation of the generation, transport and interaction of a generic gust with the DLR-F15 airfoil.

VII. Conclusion and outlook

The results in the present paper describe an ongoing cooperative effort of the DFG Research Unit 1066 to simulate wing stall. The research aims at including the effects of gusts that distort the onset flow. The present paper serves to disseminate the results to the international audience. The currently obtained data on wing stall is publically open and can be shared with other research groups who seek cooperation on the subject. The ongoing research of the Research Unit will continuously improve the physical models, refine the computational algorithms and broaden the experimental data base over the next years.

References

[1] www.for1066.de , October 2012.

[2] Niehuis, R.; Lesser, A.; Probst, A.; Radespiel, R.; Schulze, S.; Kähler, C.; Spiering, F.; Wartzek, F, Schiffer, H.-P.: "Simulation of Nacelle Stall and Engine Response". 21th ISABE Conference, 9-13 September 2013, Busan, Korea, 2013.

[3] Cecora, R.-D., Eisfeld, B., Probst, A., Crippa, S., Radespiel, R.: "Differential Reynolds Stress Modeling for Aeronautics". AIAA-2012-0465, 2012

[4] Jakirlic, S., Hanjalic, K.: "A new approach to modelling near-wall turbulence energy and stress dissipation". Journal of Fluid Mechanics, 459(1), Cambridge Univ Press., 2002, pp. 139-166.

[5] Probst, A., Radespiel, R., Rist, U.: "Linear-Stability-Based Transition Modeling for Aerodynamic Flow Simulations with a Near-Wall Reynolds-Stress Model". AIAA Journal, 50(2), American Institute of Aeronautics and Astronautics, 2012, pp. 416-428.

[6] Probst, A.; Schulze, S.; Kähler, C.J.; Radespiel, R.: Reynolds-Stress Modelling of Subsonic and Transonic Inlet Stall Compared to Measurements. Third Symposium of Wing and Nacelle Stall, 21-22 June 2012, Braunschweig, Germany, www.for1066.de/

[7] Knopp, T., Probst, A.: "Numerical Simulation of Engine-Inlet Stall with Advanced Physical Modelling Compared to Validation Experiments". To appear in: New Results in Numerical and Experimental Fluid Mechanics VII. Contributions to the 17th STAB/DGLR Symposium Berlin, Germany 2010.

[8] Raasch, S., Etling, D.: "Modeling deep ocean convection: Large eddy simulation in comparison with laboratory experiments". J. Phys. Oceanog., 28, 1786 - 1802, 1998.

[9] Raasch, S., Schröter, M.: "PALM - A large-eddy simulation model performing on massively parallel computers". Meteorol. Z., 10, 363-372, 2001.

[10] Camp, D.W.: "Wind velocity measurements of low level wind gust amplitude and duration and statistical gust shape characteristics ". National Aeronautics and Space Administration, Technical Memorandum, 1968.

[11] Bange, J., Spieß, T., van den Kroonenberg, A., „Characteristics of the early-morning shallow convective boundary layer from helipod flights during stinho-2". Theor. Appl. Climatol. 90, 113–126, 2007.

[12] Auerswald, T., Bange, J., Knopp, T., Weinman, K. and Radespiel, R.: "Large-Eddy Simulations of realistic atmospheric turbulence with the DLR-TAU-code initialized by in situ airborne measurements". Computers & Fluids, 66, 121–129, 2012

[13] Heinrich, R. und Reimer, L. Michler, A.: "Multidisciplinary simulation of maneuvering aircraft interacting with atmospheric effects using the DLR TAU code". RTO AVT-189 Specialists' Meeting on Assessment of Stability and Control Prediction Methods for Air and Sea Vehicles, 12.-14. Oktober 2011, Portsdown West, UK.

[14] Jirasek, A.: "CFD Analysis of Gust Using Two Different Gust Models". RTO AVT-189 Specialists' Meeting on Assessment of Stability and Control Prediction Methods for Air and Sea Vehicles, 12.-14. Oktober 2011, Portsdown West, UK.

[15] Spiering, F.; Kroll, N.: Coupling of TAU and TRACE for parallel accurate flow simulations. International Symposium „Simulation of Wing and Nacelle Stall", Germany, Braunschweig, 21st - 22nd June 2012, www.for1066.de, October 2012.

[16] Lacor, C., Smirnov, S., Baelmans, M.: A finite volume formulation of compact central schemes on arbitrary structured grids. Journal of Comp. Physics 198, 535—566, 2004.

[17] Hahn D., Scholz P., Semaan, R., Radespiel R., Mueller-Eigner R. "Erzeugung definierter Querwirbel in einer geschlossenen Messstrecke" 60. Deutscher Luft- und Raumfahrtkongress, 27. - 29. September 2011, Congress Zentrum Bremen, 2011.

[18] Hahn D., Scholz P., Radespiel R. "Vortex generation in a low speed wind tunnel and vortex interactions with a high-lift airfoil" AIAA-2012-3024, 30th AIAA Applied Aerodynamics Conference, New Orleans, Louisiana, June 25-28, 2012.

[19] Reuss, S.; Probst, A.; Knopp, T.: Numerical investigation of the DLR F15 two-element airfoil using a Reynolds stress model. Third Symposium of Wing and Nacelle Stall, 21-22 June 2012, Braunschweig, Germany, www.for1066.de/