Simulation of pressure and shock induced separation using DES implementations in the DLR-TAU Code

Sebastian Illi,*
Thorsten Lutz
Ewald Krämer
Institute of Aerodynamics and Gas Dynamics, University of Stuttgart, 70569, Stuttgart, Germany

Abstract

Although computational fluid dynamics made its way to become an essential tool for aerodynamic development, there are regions within the flight envelope where RANS methods reach their limits. In these regions massive separations due to high pressure gradients or shocks occur. The presentation covers the simulation at these limits using DES-methods to simulate low and high speed stall on extruded airfoils. At first, an overview over the new implementations in the unstructured DLR-TAU-Code is given. Numerical results for a backward facing step are compared to experimental results and provide a validation basis for further examined cases. Especially differences in the results of DES and RANS methods are highlighted and a way to determine the percentage of turbulence kinetic energy is shown.

Because DES is very sensitive to the spatial discretization in the LES domain, different methods to estimate it are presented. These methods include the use of priori made URANS simulations to directly calculate the length scale, to determine the limits of the inertial subrange of the energy spectra or to create the model energy spectra suggested by Pope.

Finally, two airfoils at stall conditions using DES implementations in the DLR-TAU code are presented. First, the separation on a NACA0012 airfoil at high angle of attack is simulated. Second, the OAT15A profile is examined at high speed stall conditions showing shock induced separation with a fast moving shock on the upper side of the airfoil.

*corresponding author, E-mail address: sebastian.illi@iag.uni-stuttgart.de

Turbulence modelling in Computational Fluid Dynamics *(CFD)* became one of the main research topics in the last decades and is considered to be one of the most difficult problems in flow physics. Many different approaches have been developed. On the one hand the Reynolds Averaged Navier-Stokes approach *(RANS)* at this time is the standard for CFD application in industry. The approach is able to capture the main regime of the flight envelope and gives acceptable results if the user is interested in the mean coefficients of attached or weakly separated flows. On the other hand there is Large Eddy Simulation *(LES)*, where the spatial and temporal resolution is very high, so that about 80% of the turbulent kinetic energy can be resolved. This method assures that the large turbulent structures are directly resolved while the scales smaller than the filter length are represented by a subgrid scale model *(SGS)*. For detached flows at high Reynolds numbers, e.g. on wings in operating conditions at the border of the flight envelope, the so-called hybrid methods offer a solution to the gap between RANS and LES methods. In areas near the wall where the small scales are dominat, the RANS equations are solved. On the other hand, if it comes to detachment of the boundary layer, the LES areas are able to resolve the large eddies to predict the unsteady characteristics of a flow.

For the following studies the flow solver TAU (Version 2010.1.0) was used. The TAU code originates from the MEGAFLOW project and is constantly developed by the German Aerospace Center DLR. TAU is a second order finite volume solver based on a cell vertex formulation, which solves the three dimensional, instationary and compressible Navier-Stokes equations. Different central and upwind schemes are used for the discretization of the inviscid and viscous fluxes. For temporal discretization Runge-Kutta and Backward Euler Schemes are implemented and for time accurate flows a the Jameson-Schmidt-Turkel scheme *(JST)* can be used. Unstructured or hybrid grids are supported, which show their advantages on complex configurations and allow grid adaptation to increase the spatial grid resolution locally which improves the accuracy especially in presence of large pressure gradients. Different one- and two- equation turbulence models and some Reynolds stress models have been implemented. TAU is also capable of using different approaches for Detached Eddy Simulations *(DES)*. In the following studies mainly the Spalart Allmaras model in combination with the DDES and IDDES approach has been used. To show the influence of the turbulence model, a comparison with the Menter-SST model has been performed.The code has been optimized for massive parallelization and was compiled for the following studies on the BWGRID cluster at the High Performance Computing Center in Stuttgart (HLRS) and on the HLRB2 at the Leibniz Data Center (LRZ) in Garching.

To get reasonable results, a mesh with a sufficient spatial resolution is required. Three methods to estimate the required resolution Δ have been investigated during the first year of the ComFliTe project and are based on the estimation of the turbulence length scales with a previously RANS simulation. Therefore a RANS simulation using a two equation turbulence model should be perfomed to get a distribution of the turbulence kinetic energy and of the dissipation where the turbulence length scale distribution can be derived from. A first approach for shock or pressure induced separation is to choose k and ω close to the separation point and to determine a length scale from dimensional analysis $L = k^{\frac{3}{2}}/\epsilon$. For high Reynolds numbers R_λ where λ is the Taylor microscale, a correlation between L and the integral length scale L_{11} exists where $L_{11} = 0.4L$ [5]. L_{11} is known as the average size of the most energy containing eddies, which has to be resolved in the LES area. Therefore $\Delta = L_{11}$ seems to be reasonable.

The second approach is based on the estimation of the limits of the inertial range of the Kolmogorov energy cascade. Following [3] the borders for the initial subrange of the spectrum k_f and k_d can be calculated, if the length scale of the most energy containing eddies and the dissipation rate are known. These values can be gathered again from a previously performed

RANS simulation. k_c is obtained by choosing a wavenumber within the inertial subrange, e.g. in the logarithmic center of k_f and k_d.

For the last approach presented here, also the theory of Kolmogorov is used. For Large Eddy Simulations it is common practice to resolve around 80% of the turbulence kinetic energy. Therefore a model energy spectrum has to be found as described in [5]. K_t can be obtained by integrating the energy spectrum $E(k)$ over all wavenumbers k. Setting the upper integration limit as k_c and solving the integral equation for $k_{res} = 0.8K_t$, k_c can be calculated.

All approaches however are based on priori RANS calculations. The length scale l_t is a function of x, y, z and, if an unsteady method is used, a function of t. It is left to the user, at what time and at which position of the URANS calculation, the values for K_t and ω (respectivly ϵ) are taken to estimate the spatial resolution. Therefore to verify the solution the only way is to check how much turbulence kinetic energy has been resolved in areas where the LES mode is active. For three test cases this is shown in the presentation on the Joint Symposium 'Simulation of Wing and Nacelle Stall' in 2010.

First the results for a backward facing step are presented, which is based on measurements of Driver and Seegmiller [2]. This case provides a validation basis for the DDES and IDDES methods used in further studies with the TAU code. The setup is a channel with a viscous wall on the upper and lower side of the channel. Case 1 with a deflection angle of 0° is simulated. The mesh is build as suggested in [6] using the experimental step height of h = 0.0127 m. The length of the inflow part is 4h, the length after the step of 25h. The height upstream of the step is 8h as it is described in [2] with a spanwise extension of 4h. As spanwise boundary condition periodicity is used. A Dirichlet boundary condition containing experimental data is used as an inflow boundary condition together with a pressure exit outflow at the end of the channel. The results of both, DDES and IDDES formulation, show good agreement with the measurement data.

The second test case is based on measurements of Seifert et. al. [7], where two airfoils (NACA 0012 and NACA 0015) have been investigated using active flow control to delay flow separation. The results for the NACA 0012 profile at low speed stall conditions (Re = 6e6, M = 0.3, $\alpha = 16°$), without active flow control, are presented. Three 2D simulations using the unsteady RANS (URANS) approach with various turbulence models (SALSA, Menter-SST, RSM) have been performed and are compared to a DDES simulation. It is shown, that the 2D-URANS simulations are able to detect separation at this angle of attack and that they are able to represent the unsteadiness of the flow. Contrary to these results the DDES shows also the unsteadiness of the flow but is able to give better agreement with the measurement mean pressure distribution.

The last case is based on measurement data of Jacquin et. al. [4] [1]. At Re = 3e6, M = 0.73 and $\alpha = 3.5°$ high speed buffet is present, which is induced through shock wave / boundary layer interaction resulting in rapid shock movement on the upper side of the airfoil. It is shown, that RANS methods are not able to predict the movement around the buffet onset. Detached Eddy Simulation on coarse meshes show 2D structures where the turbulence kinetic energy is not sufficiently resolved. Anyhow the method is able to predict the measured frequency and the c_{p-rsm} values even on these coarse grids. Further investigations will be performed on gradually higher resolved meshes to resolve the desired percentage of turbulent kinetic energy.

References

[1] S. Deck. Numerical Simulation of Transonic Buffet over a Supercritical Airfoil. *AIAA*, 43:1556–1566, 2005.

[2] D. M. Driver and H. L. Seegmiller. Features of a Reattaching Turbulent Shear Leayer in Divergent Channel Flow. *AIAA*, 23:163–171, 1985.

[3] J. Froehlich. *Large Eddy Simulation turbulenter Stroemungen*. Teubner, 2006.

[4] L. Jacquin, P. Molton, S. Deck, B. Maury, and D. Soulevant. An Experimental Study of Shock Oscillation over a Transonic Supercritical Profile. In *35th AIAA Fluid Dynamics Conference and Exhibit*, 2005.

[5] S. B. Pope. *Turbulent Flows*. Cambridge University Press, 2000.

[6] A. Probst, R. Radespiel, Ch. Wolf, T. Knopp, and D. Schwamborn. A Comparison of Detached-Eddy Simulation and Reynolds-Stress Modelling Applied to the Flow over a Backward-Facing Step and an Airfoil at Stall. In *48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, 2010.

[7] A. Seifert and G. Pack. Oscillatory Excitation of Unsteady Compressible Flows over Airfoils at Flight Reynolds Numbers. In *37th AIAA Aerospace Sciences Meeting and Exhibit*, 1999.