Local boundary layer resolving large eddy simulations for hybrid IDDES methods

Behavior of local boundary layer resolving large eddy simulations in low pressure turbine flows using zonal hybrid methods such as IDDES

***H. Seehausen, J. R. Seume**, Institute of Turbomachinery and Fluid Dynamics, Leibniz University Hannover*

In Short

- Improve turbomachinery design
- Gain experience with hybrid CFD methods
- Develop numerical experiments

In this project, the use of hybrid methods, such as Improved Delayed Detached Eddy Simulation (IDDES), will be investigated in the context of flow around low-pressure turbine blades (LPT). Under the concept of zonal-hybrid scale-resolved simulations (SRS) methods such as IDDES, the boundary layer regions are calculated in Reynolds-Averaged Navier-Stokes (RANS) mode and the free shear layers are calculated in large eddy simulation (LES) mode. Accordingly, turbulence and transition are fully modeled in the boundary layers (RANS) and calculated to a predominant extent outside (LES). The differentiation between RANS and LES mode is controlled by the mesh resolution. If this is sufficiently fine, the LES mode is activated. In preliminary work at MTU Aero Engines AG (MTU, formerly Motoren- und Turbinen-Union), it was found that the sidewall boundary layers and secondary flow regions in LPT cascades cannot be modeled with sufficient accuracy using the RANS approach. Therefore, it will be examined whether the model can be forced into LES mode by appropriately adjusting the mesh resolution in the region of the sidewall boundary layers and whether this will improve the currently deficient predictions. Simulations are performed using the flow solver TRACE on the MTU T161 and T170 turbine blades. But what is the benefit of hybrid methods?

A key component of aircraft that plays an important role in fuel consumption are the engines and their related efficiencies. The turbine and compressor stages are important elements that greatly influence this figure. Their blades and vanes can basically be thought of as simple airfoils. Traditionally, these are tested in a wind tunnel with a simplified design called a cascade, where some of these blades are arranged in a row, as opposed to the typical circular design of jet engines, where the duct geometry remains the same. Instrumental probes can then be used to study the various performance characteristics of the design. Such experiments have the major disadvantage of being costly and slow to perform. In addition, the data are obtained at a few discrete positions and the probes in question can potentially interfere with the measurements. Computational fluid dynamics (CFD) offers an alternative in which the full three-dimensional unsteady flow can be studied with a theoretically unlimited degree of resolution. The entire flow field can be resolved and a nearly continuous set of measurements can be obtained, allowing for an improved design.

Unfortunately, this all remains theory, and practice has not yet caught up. Such simulations are possible for standard flows only with Direct Numerical Simulation (DNS), which solves the entire flow directly using the Navier-Stokes equations without additional modeling. However, DNS is far too computationally expensive to be performed on modern machines. In the short term, RANS simulations remain the de facto industry choice when it comes to computational fluid dynamics (CFD), as it is fast and computationally inexpensive. However, they suffer from a lack of accuracy due to their modeling nature. On the other hand, SRS such as LES provide an increase in accuracy, which is at the cost of a much higher computational effort. To remedy this, hybrid methods have been proposed, attempting to combine the advantages of both models while overcoming their weaknesses.

With respect to the question, this means that hybrid methods can be used for numerical experiments as well as for design if they are sufficiently proven. Therefore, the objective of this project is to investigate IDDES in the context of turbine blades and to compare it with experimental results and the RANS model currently used in the industry. This is done using the Turbomachinery Research Aerodynamic Computational Environment (TRACE) flow solver [1] developed by the German Aerospace Center (DLR, Deutsches Zentrum für Luft- und Raumfahrt) and internally at MTU Aero Engines AG (MTU, formerly Motoren- und Turbinen-Union). The results will be published to share the experience with the scientific community. Consequently, this

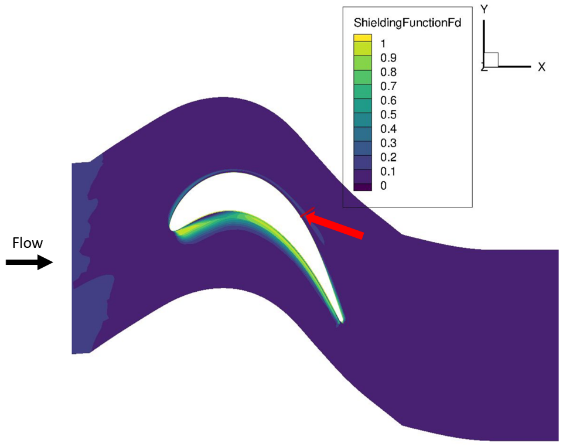

Figure 1: *IDDES shielding function $\bar{f}_d$. $\bar{f}_d$ = 0 indicates LES behavior, whereas $\bar{f}_d$ = 1 indicates RANS behavior.*

leads to a better understanding of the models and allows an ongoing development and improvement of these methods.

A first investigation of a simplified geometry of a turbine cascade was performed to analyze parameters specific to hybrid methods. In Fig. 1 a time average of the shielding function, which indicates the method of simulations (RANS or LES) is depicted as a blade-to-blade view. In the boundary layer region close to the blade, it can be seen that the shielding function is activated and becomes $\bar{f}_d$=1. This means that in this area the RANS equations are used. On the pressure side (bottom side of the blade), a large area with RANS resolved flow is formed. Here is a pressure side separation of the flow. The ratio of local flow to cell size is not sufficient for LES in this region. The remaining area of the flow region is resolved with a LES.

In this project, this investigation will be extended to 3d full models of the blades T161 and T170. Local changes in mesh resolution will be used to control the simulation method used (RANS or LES). This will provide insight into the required mesh resolution in different regions of the computational domain. Consequently, the two questions below should be adressed by the project: Which influence does this changeable behavior have on the convergence of the solution compared to a pure LES? Is it possible to force the calculation of the sidewall boundary layer into the LES mode to benefit from the more accurate LES solution when using hybrid methods under consideration of consumable computational resources?

WWW

https://www.tfd.uni-hannover.de/en/

More Information

[1] DLR, *TRACE user guide: overview*, Retrieved October 15, 2022, from http://www.trace-portal.de/userguide/trace/index.html,

DFG Subject Area