Numerical analysis of integration of nacelle to a high lift aircraft configuration

Installed adVAnced Nacelle uHbr Optimisation and Evaluation

***G. Subbian, R. Radespiel**, Institute of Fluid Mechanics, TU Braunschweig*

In Short

- Multi-point optimization of installed nacelles for UHBR engines,
- Transonic wind tunnel tests,
- Validation of optimized installed nacelle performance.

Since the operation of turbofan engines for commercial airplanes, both the air traffic as well as the efficiency of aircrayt engines have lineraly increased. An efficient aircraft operation model is highly important and it leads to sustainable aviation by reducing the carbon footprint. Fuel prices are steadily increasing and industries are looking at alternative options to reduce the dependancy on fossil fuels. Neverthless, airlines can get better at managing operational costs by reducing the fuel consumption resulting in reduced burden on the passangers as well as the industry. It has been prognosed that the yearly increase of air travel will be around 4 - 6% [1] and strategies to reduce emissions should be strongly pursued more so than ever, given the adverse effects of global warming for all life forms. Most of the modern turbofan engines operating currently have a ByPass Ratio(BPR) of around 12:1 and it is forseen that the future Ultra High ByPass Ratio(UHBR) would be in the region of around 12-20:1 [1]. This results in larger and efficient aircraft engines. These engines are housed in the nacelles and attached to the aircraft wings at suitable positions. The design and development of advanced nacelles should also keep up in pace, parallel with the development of UHBR engines. European Union (EU) has always placed high emphasize on working towards stopping the human induced climate change. Through the efforts of clean sky program, various programs in diverse domains are supported towards climate friendly aviation. The combined european project "Installed adVAnced Nacelle uHbr Optimisation and Evaluation"(Project ID: 863415, Call ID: H2020-CS2-CFP09-2018-02) is a joint effort towards the design an development of an advanced nacelle configuration for the future UHBR turbofan engines. The project partners are from university as well as industry, resulting in a close co operation between numerical design and development leading to testing in the wind tunnel as well as validation with high fidelity CFD tools.

The IVANHOE concept provides a methodology to optimise nacelle location and geometry. Ultimately, the methodology will be able to support a concurrent design process of wing, engine and nacelle which will allow to reach a global optimum in thrust/drag/lift performance. The project concept is based on three major streams of activities:

- Multi-objective, multi-point optimization of installed nacelles for UHBR engines,
- Transonic wind tunnel tests on optimized nacelles including advanced measurement techniques,
- Validation of optimized installed nacelles performance and high-lift investigations.

The first and third streams need reliable numerical flow simulations to achieve their objectives. The numerical simulation approach of the IVANHOE project takes advantage of carefully selected and qualified numerical methods with models of turbulence beyond the state of the art. For reasons of computational efficiency, the partners employ the robust and numerically efficient commercial code Fluent during automated optimization processes, however with a careful checkout of its numerical accuracy by using cross comparisons against the aeronautical flow solver DLR-TAU. The DLR-TAU flow solver offers the opportunity to use advanced simulation models beyond the state of the art in comprehensive CFD analyses. These are Reynolds-stress models of turbulence (RSM) that have been validated for aircraft applications at high-Reynolds numbers during previous works by the German Aerospace Center, DLR, and TU Braunschweig. These models will be employed to represent the effects of secondary flows in nacelle and pylon junctions in transonic flow, and they aim at resolving longitudinal vortices at take-off flow conditions much better than with eddy-viscosity models. The IVANHOE project will build on these technology advancements to bring CFD evaluation of installed UHBR nacelles to a new level. To achieve these project objectives, dedicated flow simulations were performed during the project. The project partners perform initial validation computations on a range of well-known and publically available test cases that represent the installation effects of bypass ratio engines on commercial aircraft. The existing public data bases of previous AIAA Workshops on drag prediction and high lift prediction serve these needs. The results of the computations by the IVANHOE partners will provide the needed evidence, that numerical simulation data are not biased by significant numerical errors or by employing unsuited or improperly implemented models of turbulence.

Common Research Aircraft (CRM) model is a generic configuration of a passenger aircraft which was extensively investigated in the AIAA Drag prediction workshops. The fuselage is similar to that of a wide-body commercial transport aircraft. It includes a wing-body fairing to accommodate the wing root and a scrubbing seal for the horizontal tail, to facilitate the variation of the tail incidence angles although the geometry considered here are Wing-Body and Wing-Body Nacelle-Pylon configurations. CRM is a low wing configuration designed to cruise at a Mach number of 0.85 with a nominal lift coefficient of 0.5. Due to the availability of extensive database and suitability for our test application, CRM has been chosen as the test case, in which the nacelle development and integration will be performed.

After performing mesh independency and turbulence model studies, higher difference in the drag values were seen between the Fluent and DLR-TAU code results. A detailed study was undertaken to find the source of this difference between the project partners. Initially not just the force coefficients were compared, but also the surface pressure distribution between the partners were analysed as shown in the Figure 1. As it can be seen, no significant differences could be observed from the wing surface pressure distribution which might point to the origin of the difference. Similar surface pressure analysis was also undertaken along the z- component of the wing to observe the stagnation region in detail. The detailed areas of lift and drag build up could be seen but no significant differences could be observed around the stagnation region between the two different flow solvers. A detailed analysis of the pressure drag and viscous drag of all the components for both the wing-body and wing-body-nacelle-pylon configurations were carried out. For the sake of reference, results from DLR computations (Spalart Allmaras and SSG/LRR-ω Reynolds stress turbulence model [2]) were also used from the AIAA drag prediction workshop. It was observed that slight drag increments to Fluent stem from all the components which finally contribute to the total drag coefficient. To solve this problem, the option of ?Warped-Face Gradient Correction? was suggested and used by HIT 09 SRL which corresponds to the hit_02 results in Figure 1. This methodology reduces the drag difference and a better agreement between the flow solvers can be seen. It is concluded by the project partners to use this option for future computations. Moreover, none of the TU Braunschweig computations shows any flow separation in the fuselage wing junction whereas the Fluent flow solver shows a small cross-flow separation. Another aspect of interest is the flight phase during take-off and landing. Numerical studies involving high lift CRM model is the current area of focus.

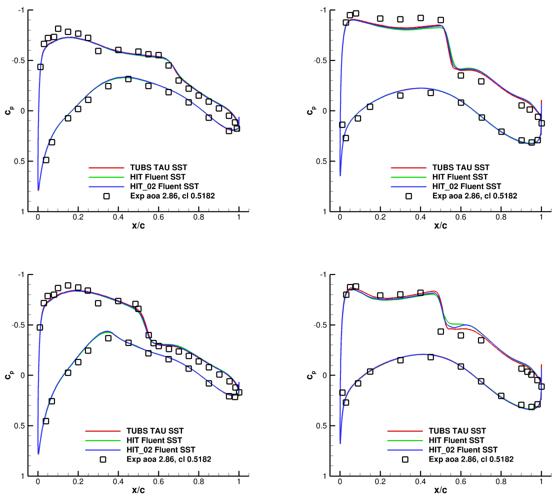

Figure 1: *Surface pressure distributions, Top: Wing body at 13% (left) and 50% (right) of wing span. Bottom: Wing body nacelle pylon at 13% (left) and 50% (right) of wing span.*

WWW

https://www.tu-braunschweig.de/ism

More Information

[1] **Merkl,E., MTU Aero Engines Deutschland:** *UHBR Aero Engines, Technologien fuer die naechste Triebwerksgeneration (ENOVAL).* Deutscher Luft- und Raumfahrtskongress 2016, Document ID 420025.

[2] **Eisfeld, B.***Implementation of Reynolds Stress Models into the DLR-FLOWer Code, Institutsbericht, DLR-IB 124-2004/31, Report of the Institute of Aerodynamics and Flow Technology, Braunschweig, ISSN 1614-7790, 2004.*

Project Partners

Chalmers University, DNW, RR, Uni PD, HIT09 SRL, Deharde GmbH

Funding

EU 863415