Optimizing and validating numerical accuracy and turbulence modeling in an industrial CFD-solver developed by Saab Aeronautics

Magnus Carlsson
Division of Fluid Dynamics, Department of Mechanics and Maritime Sciences
Aerodynamics Department, Saab Aeronautics

October 31, 2022

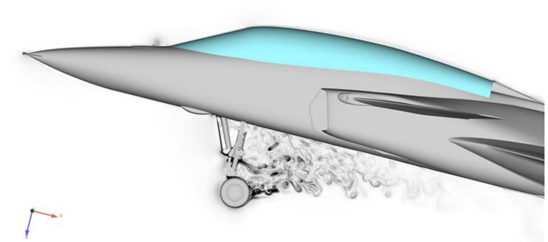

Figure 1: Density gradients of resolved turbulence behind nose landing gear. Used with permission by Saab.

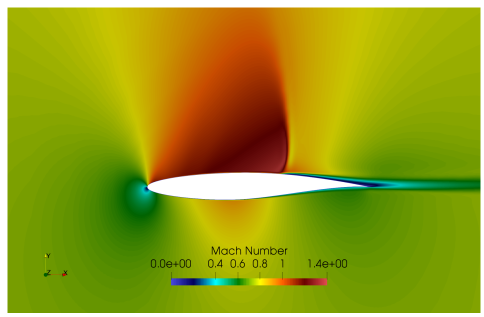

Figure 2: Mach number contours indicating shock wave pattern and flow separation over RAE 2822 airfoil.

Background

The aerospace industry needs to reduce their product development and life cycle costs, reduce the environmental footprint related to the products and increase the product capability and availability to meet customers' and consumers' needs. By introducing flow simulations techniques which are able to accurately simulate complex compressible flows, there is a great potential to reduce costs for wind tunnel and flights tests. An example were advanced flow simulations have been used can be seen in Figure 1, which presents resolved turbulence behind a nose landing gear illustrated by density gradients.

Computational Fluid Dynamics (CFD) and Reynolds-Averaged-Navier-Stokes (RANS) simulations are powerful tools in the aeronautical industry. Through simulations it is possible to predict crucial properties already in the early phases in the aircraft design, such as aerodynamic drag and forces.

The predictions generated from CFD are heavily dependent on the numerical scheme of the flow solver and choice of turbulence model. In supersonic flow, which often occurs in aeronautical applications, shock waves will be present as shown in Figure 2. The predicted location of the shock wave and the separated flow region behind it is a complex interaction between numerical scheme and turbulence modeling. Aerodynamicists need a deep understanding of these complex phenomenon in order make sound and justified decisions in aircraft design and development.

Methodology

The students will use the CFD-solver M-Edge, used and developed by Saab, to investigate the impact of the numerical scheme and the choice of turbulence model for fundamental flow cases.

- The students will evaluate and compare the numerical scheme formulation with the Spalart-Allmaras (SA), Menter Shear Stress Transport (SST) and EARSM turbulence models.
- The students will evaluate the aforementioned methods on various fundamental flow cases such as in zero pressure gradient boundary layer flow (ZPGBLF), subsonic flow over an aerofoil (NACA0012), transonic flow over an aerofoil (RAE2822) and the flow over a backward facing step (BFS). Additional flow cases can be suggested by the students as well.
- Computational grids can be provided or generated in Ansys Icem CFD by the students.

Objective

The objective of the bachelor thesis is to

- generate a deeper understanding for fluid mechanics in aeronautics,
- create a database of validation results for fundamental flow cases relevant for aeronautical applications. Results such as predicted drag and lift, location of shock wave and separation region will evaluate the performance of the models,
- if time permits, apply the analyzed numerical scheme and turbulence models in more complex flow cases, such as for a wing at high angle of attack.

The results and the validation database of the fundamental flow cases will be used in the ongoing development of the flow solver M-Edge used by Saab.

Outcome

The students will have a great opportunity to get familiar with and solve complex problems faced by the aeronautical industry. Moreover, the students will get insight into the flow solver used by Saab Aeronautics and an understanding for the CFD process from geometry handling and grid generation to evaluated flow results. With knowledge and experience from the proposed project students are well prepared for master level projects and courses in CFD and fluid dynamics within e.g. the master programs Applied Mechanics (MPAME, fluid dynamics track) and Mobility Engineering (MPMOB, aerospace engineering track). At the end of this project, involved students are encourage to propose follow-up studies which can be executed in project courses on master level and/or in master thesis projects.

Pre-requisites

The students should have an interest in fluid mechanics and preferably taken a basic course (MTF053 - Strömningsmekanik, TME055 - Strömningsmekanik or similar). Having basic knowledge in Python is recommended.

Target Group

Teknisk fysik, Teknisk design, Maskinteknik, Teknisk matematik, Kemiteknik, Kemiteknik med fysik, or similar.

Group Size

4-6 Students

Supervisor

Magnus Carlsson, magnus.carlsson@chalmers.se

Examiner

Lars Davidson lada@chalmers.se