Lucerne University of Applied Sciences and Arts

Engineering and Architecture
BSc. Energy and Environmental Systems Engineering
Bachelor-Thesis

APPLICATION AND ASSESSMENT OF GPU-BASED FLOW SOLVERS FOR TURBULENT FLOWS

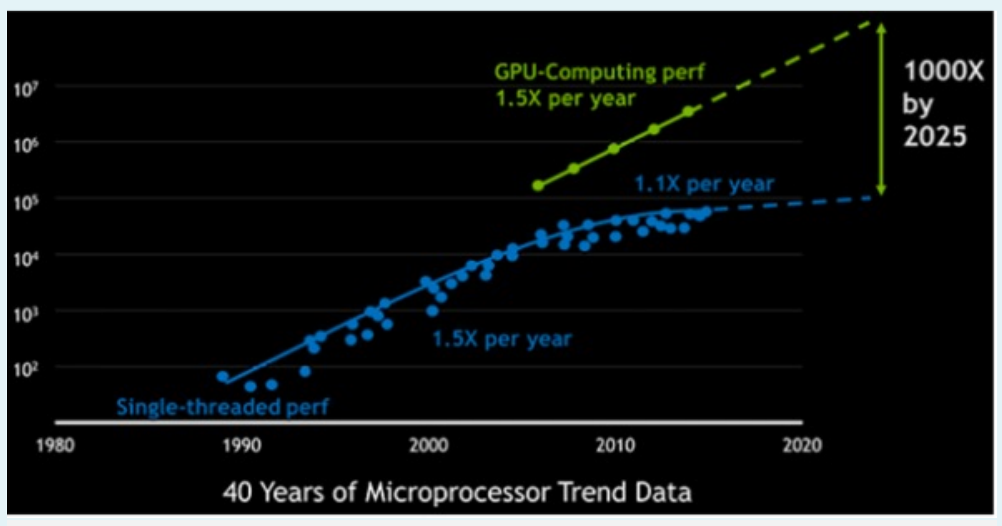
Student: Stefan Mauchle

Practice Partner: Competence Center of Fluid Mechanics and Numerical Methods
Fluid Mechanics is omnipresent! The experts at the Competence Center Fluid Mechanics and Numerical Methods optimize the flow conditions in a wide variety of products using simulations and experimental tests.

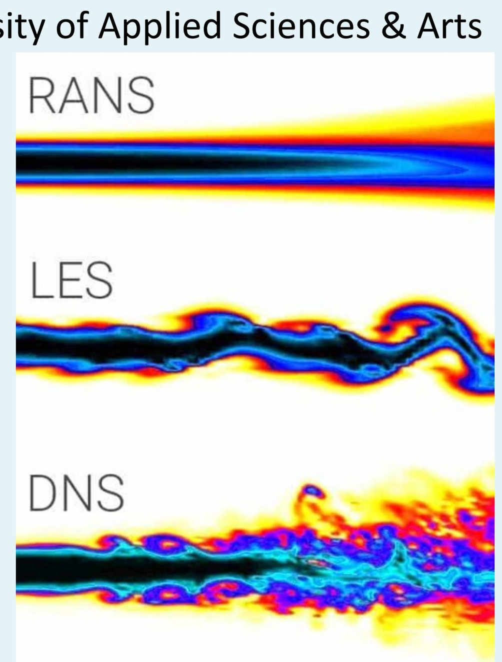
1. Background, Challenge & Objectives

Purpose

The principal objective of this Bachelor Thesis, titled "Application and Assessment of GPU-based Flow Solvers for Turbulent Flows", is to assess the capacity of GPU-based CFD solvers, specifically Ansys Fluent and TurboLab, to effectively leverage the capabilities of GPU hardware in simulations involving turbulent flows. Amidst the surge in GPU technology, these solvers have emerged as viable tools for managing complex fluid dynamics simulations. This investigation targets the crucial question: can these GPU-based solvers exploit the advances in GPU hardware to handle the challenges of turbulence models in complex flows, particularly in light of the shift towards Large Eddy Simulation (LES) methods, which traditionally have been CPU-intensive? This study aims to provide valuable insights into this critical concern and guide the Competence Center of Fluid Mechanics and Numerical Methods (CC FNUM) at the University of Applied Sciences & Arts in their selection and development of future CFD solvers.



(*Yahoo Is Part of the Yahoo Family of Brands*, n.d.)

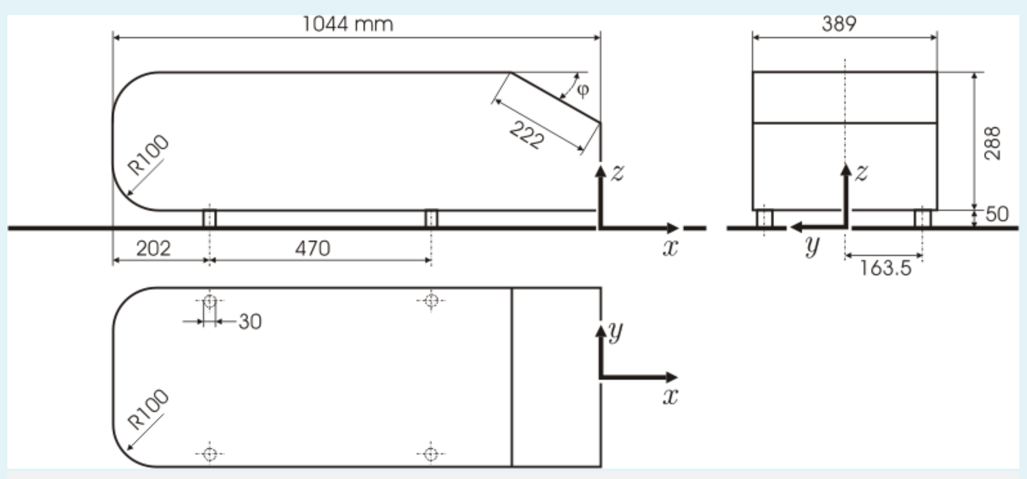


Difference RANS, LES and DNS (Gestione, 2022)

2. Methodology / Materials

Methodology

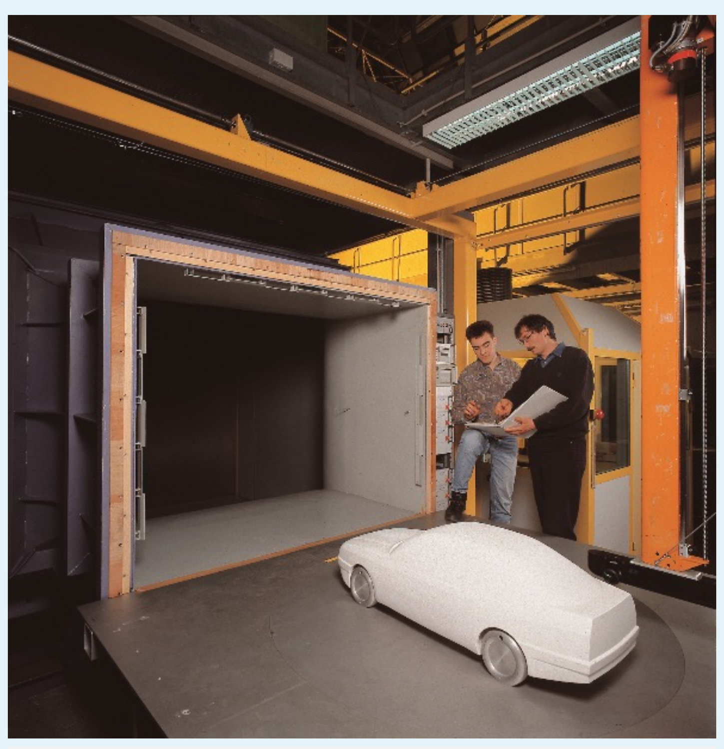
The methodology chapter outlines the approach for assessing GPU-based flow solvers for turbulent flows, focusing on the Ahmed body case. The Ahmed body, a standard model for studying vehicular aerodynamics, was selected for its complexity and available experimental data. The chapter provides detailed setups for simulations using three different Computational Fluid Dynamics (CFD) solvers: Fluent, TurboLaB, and PowerFLOW. Fluent and TurboLaB were executed on GPU, while PowerFLOW was run on CPU. The results were scrutinized for computational errors, deviations, and convergence. The aim was to determine the most accurate and efficient tool for real-world vehicle aerodynamics simulation.



The Ahmed body SimFlow CFD

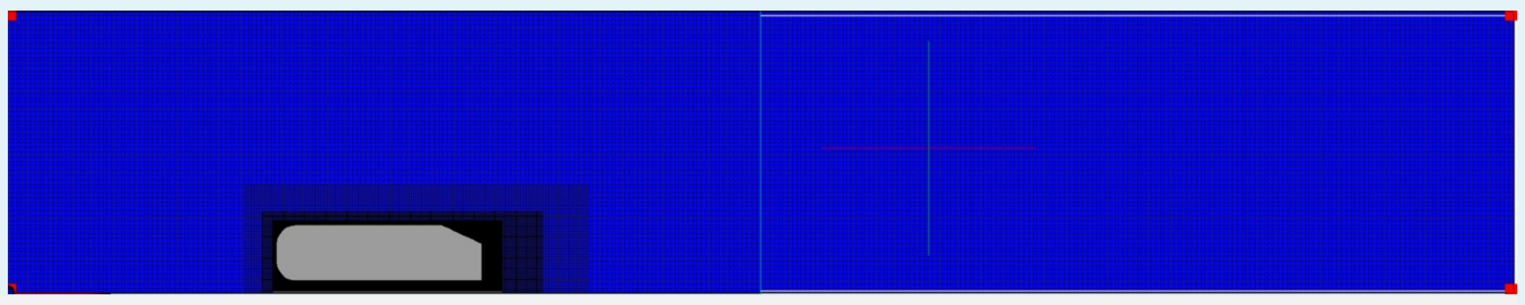
Materials / Data / Tools

- Ansys Fluent
- TurboLaB
- PowerFLOW
- Pointwise
- Excel
- Phyton
- Paraview
- GPU and CPU cluster



Wind tunnel with different model LSTM, n.d.)

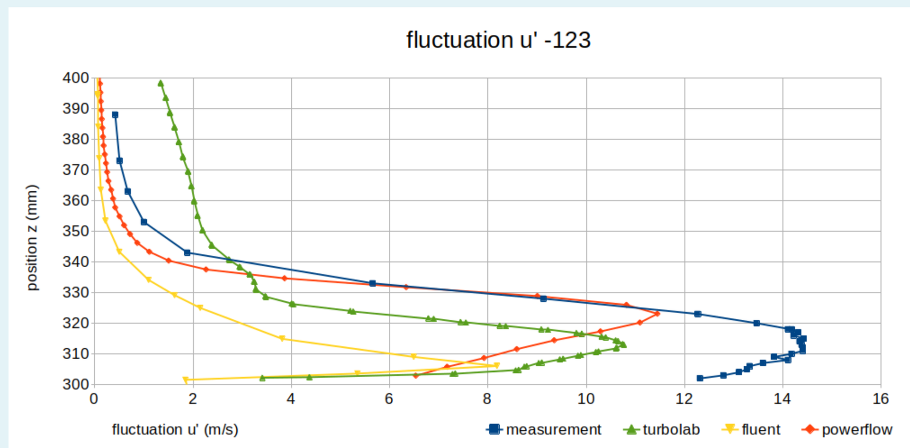
3. Results / Solution / Recommendations

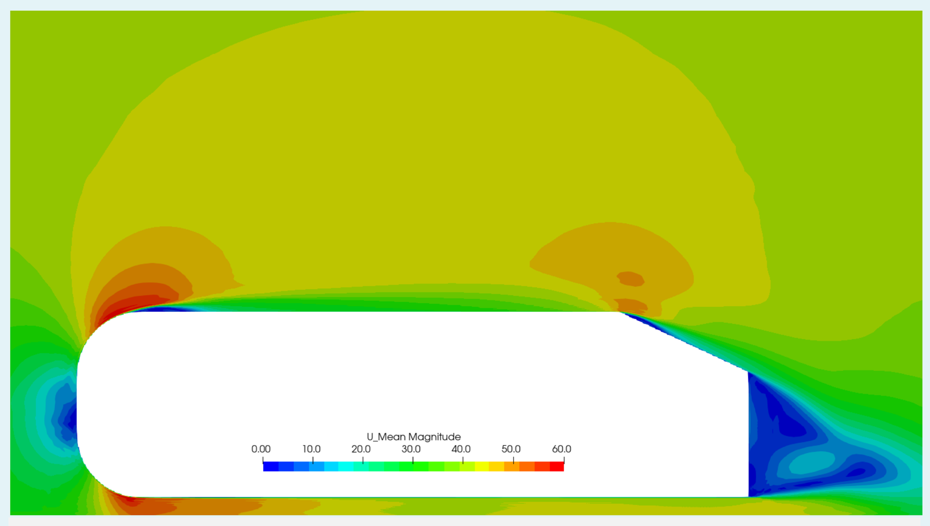


Symmetry slice, mesh PowerFLOW

Findings

Fluent offered the shortest simulation time but compromised result accuracy. TurboLaB demonstrated higher computational efficiency but lacked a functional wall function, impacting result accuracy. PowerFLOW, although slowest, provided the most accurate results. The study also exposed bugs in TurboLaB's code, revealing its potential for future development. Ultimately, the choice of solver should be task-specific, and the results offer valuable insights for future usage and development of these tools. This is due to GPU solvers not yet ready to provide both the fastest simulation time and most accurate results, but with the current rate of improvement in these codes, it is only a matter of time until this is achieved.





Flow separation

4. Discussion, Conclusions & Outlook

Overview of advantages and disadvantages

Solver	Fluent	TurboLaB	PowerFLOW
Advantages	+ short simulation time + Graphical user interface + Big community with assisting with problem solving	+ shorter simulation time compared to PowerFLOW + Fast set-up of the case + Semi-automated mesh generator	+ Most accurate results + Graphical user interface + Fully automated mesh generator
Disadvantages	- Inaccurate results - Takes more time to set-up a case compared to the other two solvers	- No turbulence model with wall function - Lack of graphical user interface - Highest RAM usage - Not commercially available - Not yet debugged sufficiently	- Longest calculation time - Small community to help with problems - Very self-contained no export functions especially for post processing - GPU not fully supported yet

Literature

Lienhart, H., & Becker, S. (2003). Flow and Turbulence Structure In the Wake of a Simplified Car Model. Retrieved March 3, 2023, from https://www.jstor.org/stable/44745411

Versteeg, H. K., & Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education.

FH Zentralschweiz

Supervisor: Prof. Dr. Luca Mangani
External expert: Joël Schlienger

Date 01.06.2023
FS23