Working Group: Prof. Dr.-Ing. habil. Thomas Indinger

Project Description:

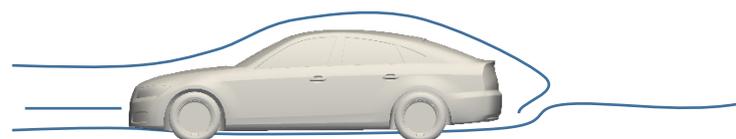M.Sc. Evelyn Rugerri
E-Mail: evelyn.rugerri@tum.de

Spilling beverages or keeping the sunroof open during a rainy day often happens by accident, which leads to an inevitable water exposure of electronic and mechatronic components in the vehicle.

The predominant physical tests are to be complemented with CFD simulations, for which the Lagrangian solver MESHFREE is utilized. Despite the flexible nature of the method, it is to be further optimized to overcome the complexities, brought by the intricate geometry.

M.Sc. Matthew Sleight
E-Mail: matthew.sleight@tum.de

Battery production today is an important topic for every car manufacturer. This research project focuses on optimizing quality control of the airtightness of batteries which is one of the longest and challenging steps in its production. This method requires air to be introduced into the battery housing under pressure to detect any leakage. The optimization of this process is made through the numerical simulation of the changing air pressure by the SPH method and experimental studies.

![TUM Logo](Lehrstuhl für Aerodynamik und Strömungsmechanik)

Prof. Dr.-Ing. Nikolaus Adams / Boltzmannstraße 15, 85748 Garching / Tel. 089/289-16138 / Fax 089/289-16139 / Homepage: http://www.aer.mw.tum.de

M.Sc. Luca Bauer
E-Mail: luca.bauer@tum.de

M.Eng. Takumi Endo
E-Mail: takumi.endo@tum.de

In the scientific world, Large Eddy Simulations have already demonstrated a good agreement with wind tunnel results. However, for industrial applications (flows with high Reynolds numbers) the computational effort is very high. Our work deals with the Wall-Modeling of Large Eddy Simulation (WMLES) for external vehicle aerodynamics using OpenFOAM. Simple validation cases, such as cylinder flow, are utilized to ensure the accuracy and reliability of the introduced modeling assumptions.

As the automotive industry advances toward decarbonization, optimizing aerodynamic performance has become crucial for reducing energy consumption. However, achieving this typically involves significant computational costs. To address the trade-off between computational expense and accuracy, this study introduces a novel wall model based on machine learning, designed to greatly reduce the grid resolution near the wall while still maintaining predictive accuracy. The wall model is developed and validated in stages. It is first created for channel flow, then adapted to flows with geometric gradients, and finally tested on the Ahmed body. The proposed method is expected to substantially lower computational costs without sacrificing physical accuracy, demonstrating strong potential for practical use in engineering simulations.

Requirements:

- Motivated, independent, and proactive approach to research.
- Familiarity with fluid dynamics and aerodynamics principles.
- Strong written and verbal communication skills in English and/or German.

Application Process:

Interested candidates should submit their applications, including a CV, academic transcripts, and a brief statement of interest. Please highlight any relevant experience.

For any questions about the project or application process, please do not hesitate to reach out. We look forward to receiving your application.