Computational Fluid Dynamics MSc

Home assignment II.
Wake of an Ahmed body

Introduction

A dream of many engineers is to be a vehicle designer. A few of them start to design vehicles in one of the university racing teams. To start to design a vehicle with the help of CFD, first, you have to know its limits.

Ahmed body is a commonly used simplified vehicle model. The primary motivation of a simulation of an Ahmed body is to examine how capable the code is to model the main flow structures around vehicles. More versions of this body can be found in the literature, from a simple bluff body to more complex geometries with tyres. For this simulation, the basic Ahmed body is used.
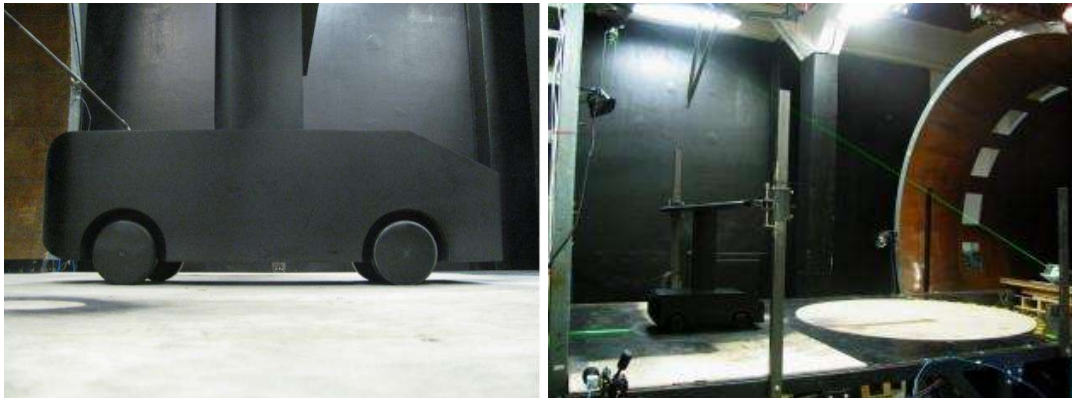

Figure 1. – Measurement of an Ahmed body with wheels at BME, AE building, Large wind tunnel [1]

Task

Simulate the flow field around a simplified car model (Ahmed Body) with a slant angle of 25° or 35°, mainly concentrating on the vehicle's wake. For validation, Flow Around a Simplified Car Body (Ahmed Body) could be used (http://cfd.mace.manchester.ac.uk/ercoftac/doku.php?id=cases:case082).

- Make a 3D half model (symmetry) according to the validation case.
- Run the steady-state simulation with 3 different turbulence models (2 equations turbulent model and at least one more complex model).
- Compare the available pressure and velocity data.
- Give suggestions for the preferable turbulence model, which one can model the wake best
- Write a technical report about your simulations.

Documentation

In the documentation, you should write down your investigation in your OWN word. On the diagrams and figures, your OWN result should be presented. The document and the calculation need to be uploaded onto [Moodle](#).

The documentation should contain the following parts:

- Aim of the simulation
- Introduction
    - Introduce the measurement
        - What and where were data measured and by which equipment
        - Uncertainty of the measured variables
    - Refer to a few papers that made a simulation for this case
        - Which type of mesh they used
        - What physical model they used
- Describe the used geometry (with figures as well)
- Describe the used mesh (with figures as well)
- Describe the physical models
- Results of the simulation with different turbulence model
    - velocity, pressure contour plots
    - y+ values
    - compare velocity and pressure plots with the measurement results
- Summary
- References

Data for validation

At least the following data should be used for the validation:

- Pressure distribution:
    - centerline (y = 0)
- LDA measurement:
    - x-z Planes: y = 0, 100, 180
    - y-z Planes: x = -88, 0, 80, 200

References

[1] András Gulyás, Ágnes Bodor, Tamas Regert, Imre M. Jánosi (2013)- *PIV measurement of the flow past a generic car body with wheels at LES applicable Reynolds number*, International Journal of Heat and Fluid Flow, Volume 43, Pages 220-232, ISSN 0142-727X, https://doi.org/10.1016/j.ijheatfluidflow.2013.05.012.