Warsaw University of Technology
Faculty of Power and Aeronautical Engineering
Computer Simulation of Combustion Processes

The Aerodynamics of Motorbike in Turbulent Flow Simulation

Author:
Krzysztof Dzedzej 299116

17.06.2021

Contents

1 Introduction

The aim of the project was to run a simulation of airflow around motorbike in OpenFOAM8 program in turbulent steady flow environment. Whatsmore the program was run on Windows with a Virtual Machine on which it was installed in ubuntu software

2 Model and initial conditions

The model and initial conditions were taken from inbuild predefined cases that were downloaded alongside OpenFOAM instalation on ubuntu. The more challenging part were the usage of different meshing model ,called snappyHexMesh which has greater complecity and is more precize than simpler blockMesh. It was used to make motorbike model more accurate and smoother ,which will result in better and more accurate results later on.

To run simulation simpleFoam was used, which is steady state solver for incompressible, turbulent flows. This solver was ideal for measuring the aerodynamics of our motorbike.

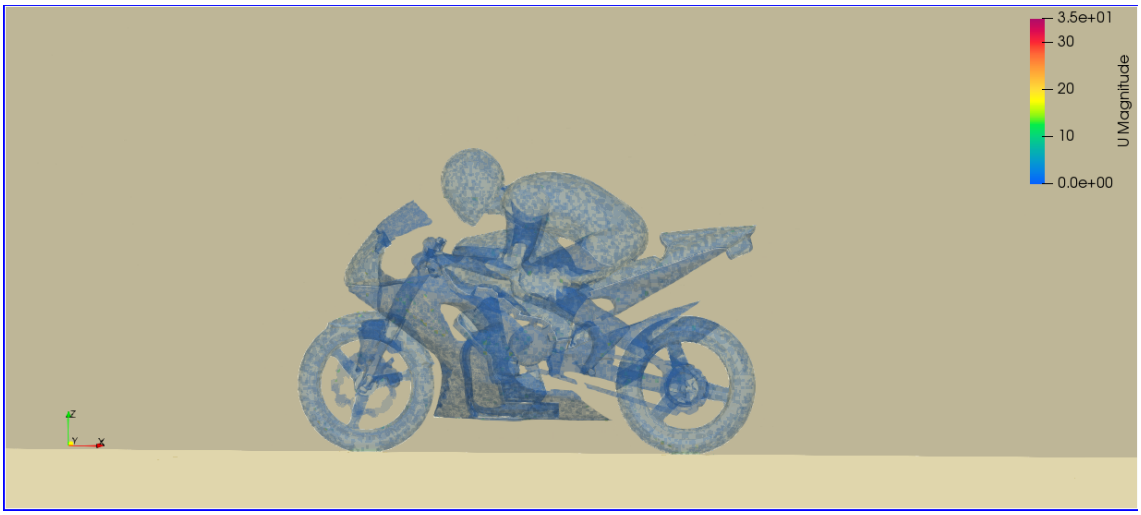

Figure 1: Mesh model

3 Results

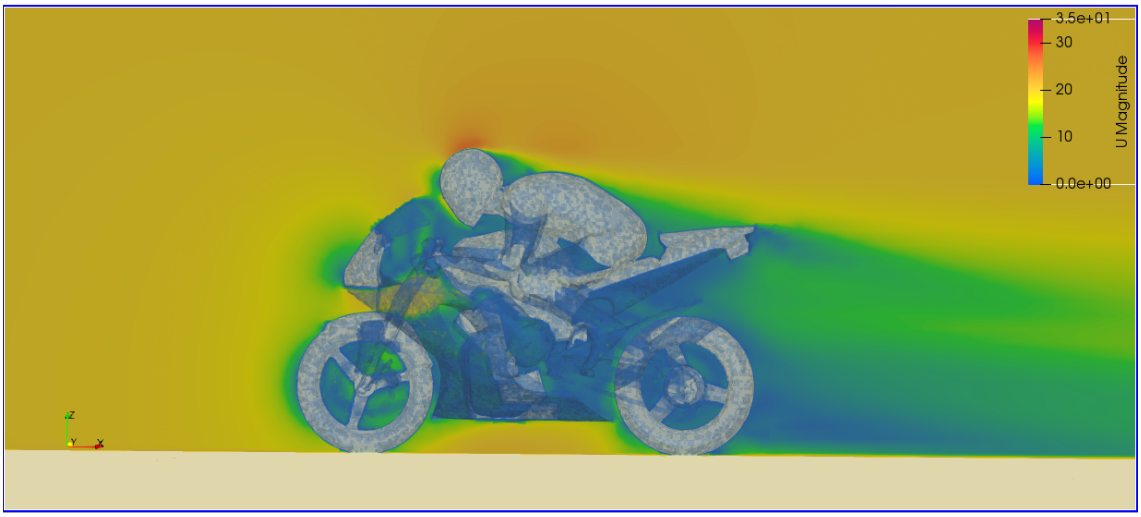

Figure 2: Velocity magnitude 1

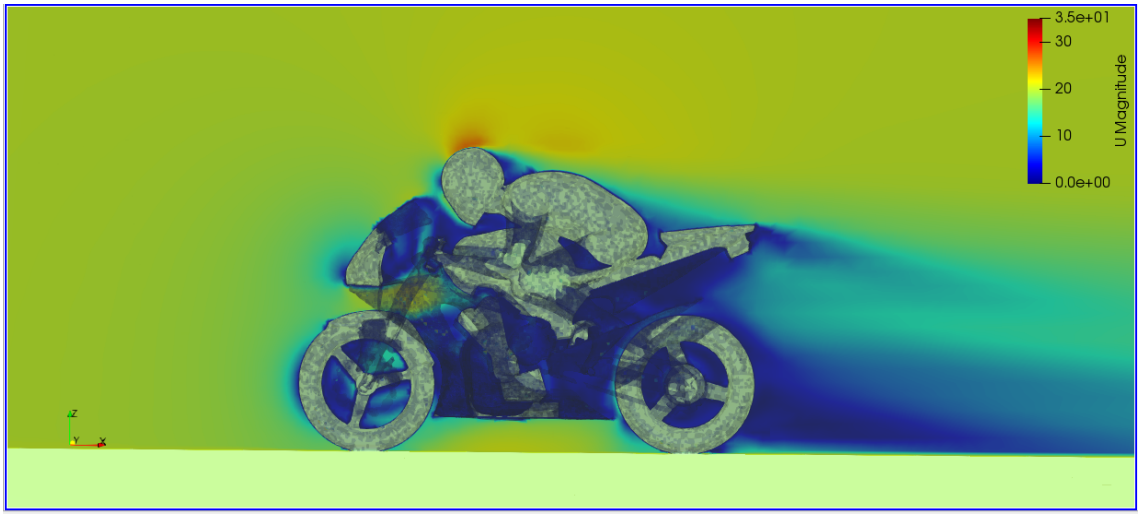

Figure 3: Velocity magnitude 2

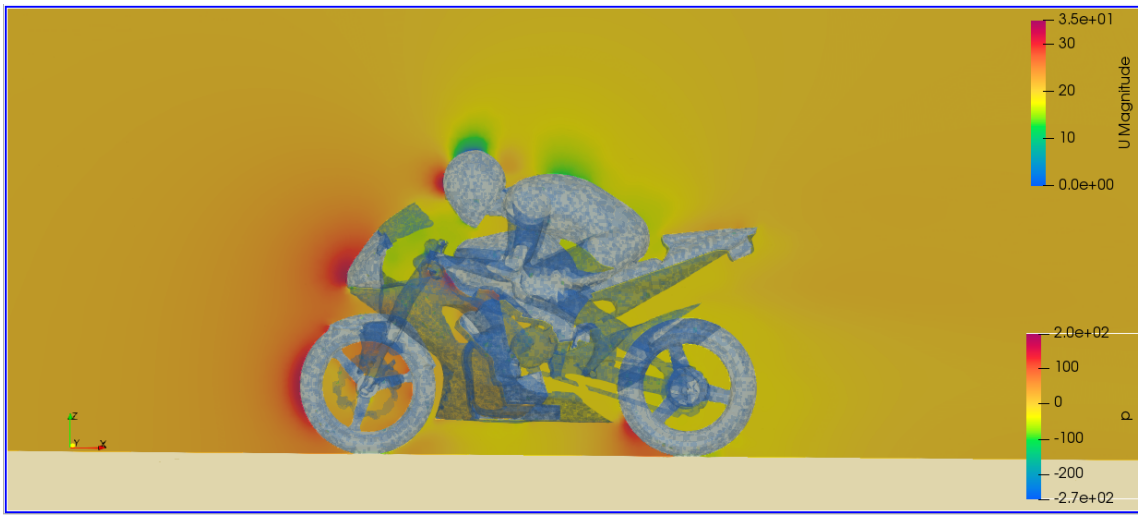

Figure 4: Pressure flow 1

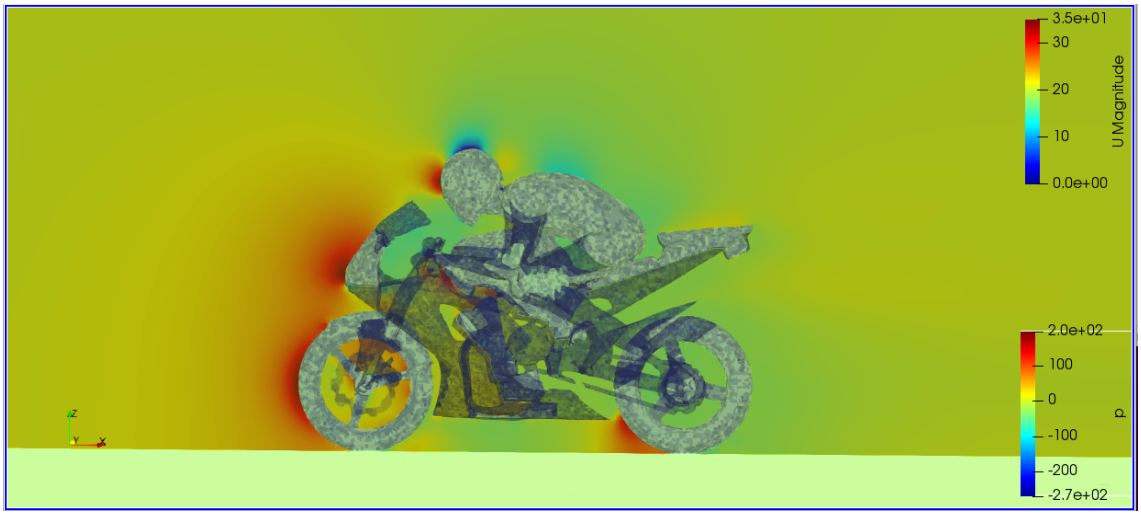

Figure 5: Pressure flow 2

4 Conculusion

OpenFOAM is really great software ,which is very useful in modeling fluid mechanics. The open source of this software enables user more flexible aproach to it in comparission to for example Ansys Fluent, however it makes it more challenging for new users and requires skills in programming to fully utilize and take advantage of this software.

It is crucial to mention that this program is free of charge for any users and is enhanced with the help of community. Additionally it nice especially for new users to find inbuild tutorials in instalation which helps them in better understandment of this software and lerning it faster.

5 Bibliography

1. $https://www.youtube.com/watch?v=sfFez7h0UUQ$
2. $https://openfoam.org/$