CFD simulation of a submarine model using Amazon Web Services

Jędrzej Śmigielski

276313

05/02/2019

1. Introduction

Aim of this project is to perform CFD simulation by parallel computing using AWS. To simulate a process of a fluid flow around simple submarine model, I have used CFD Direct From the Cloud and OpenFoam 5.2 with Ubuntu 18.04 as a operating system. After launching an instance on AWS, I needed to use Putty as a terminal, which is suitable for Microsoft Windows.

2. Case details

To create this case, I was supported by Youtube tutorials of similar cases. (https://youtu.be/uV84p_GTUPU) Unfortunately, size of its files was too big for Github, so few are not available in the repository (one of the mesh files – 50+mb) Results of all computing are presented below, pressure and velocity resolution from two different views.

3. Results

There is a visible growth of pressure at the front of submarine, as expected. Cylindric objects at the top of the body caused velocity drops, however general flow of the fluid was not significantly disturbed. Higher drop can be observed, behind top part of submarine body and at the very end, where screw propeller should be located.
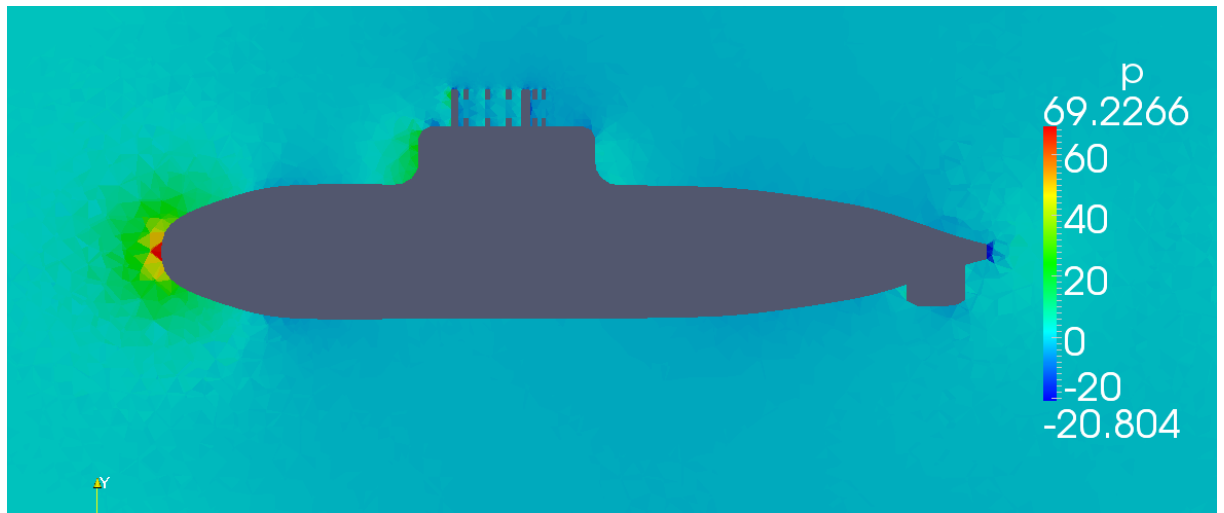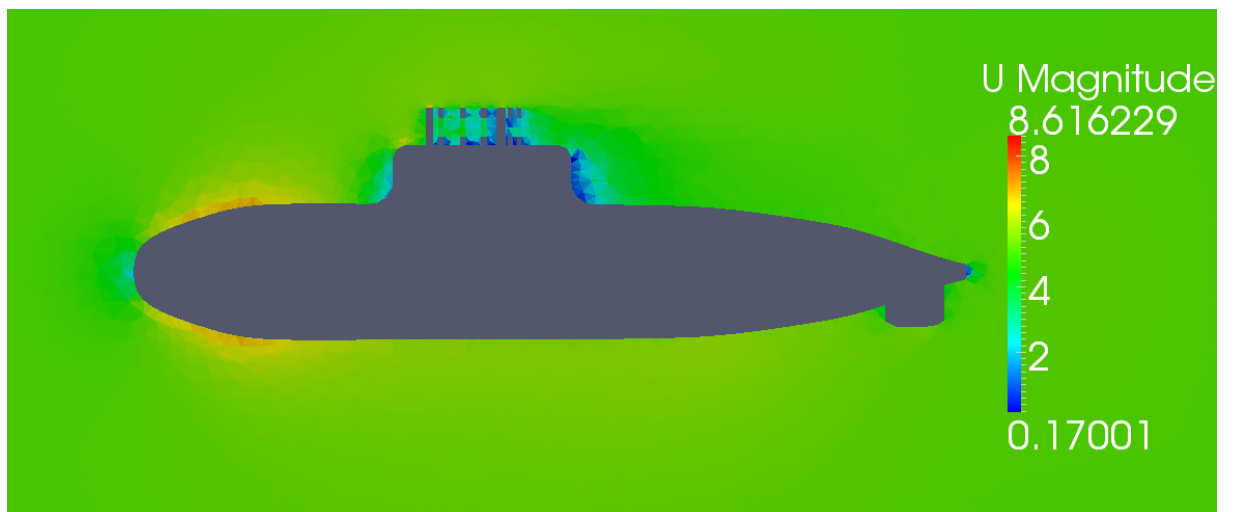
Figure 1 Pressure, side view
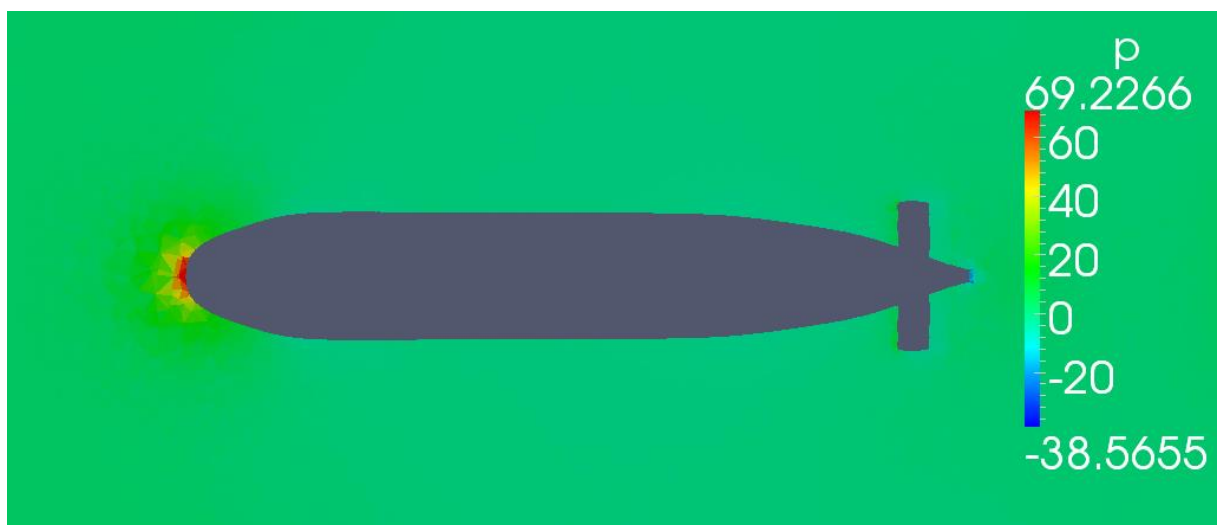

Figure 2 Velocity, side view

Figure 3 Pressure, top view

Figure 4 Velocity, top view