

*Supervisor:*

dr inż. Mateusz Żbikowski

*Authors:*

Oliwia Wacławek, 285685

Piotr Konopka, 285656

# CFD simulation of incompressible 2D flow in channel with a step

Cloud computing – project

# Contents

1. Introduction.....	3
2. Model and execution.....	3
3. Results .....	4
4. Conclusion .....	12
5. Sources .....	12

## 1. Introduction

The goal of this project was to make a simple simulation of 2D fluid flow using OpenFOAM and conducting calculations on AWS EC2 cluster rather than on a local machine. In this example a simple pitzDaily tutorial case with some modifications was used. Furthermore, simpleFoam solver was used. Another important assumption of this project was to run calculations in parallel, using all available cores/threads on multiple instances in one cluster.

## 2. Model and execution

An entire project was done via Ubuntu 18.04 virtual machine as it was far easier than using Windows based equivalents of required software. First step of the project was to install and configure CFDDFC CLI package to enable fast and easy launching and managing of EC2 instances. Below are the commands used to create a cluster of 2 machines, divide case to all processors and perform calculations:

```
cfddfc launch -i m5a.large #creates a master instance
```

```
cfddfc cluster -slaves 1 -key ***** -type demand #creates 1 slave instance, giving a cluster of 2 instances with 2 processors each
```

```
cfddfc push -key ***** #allows to send case files to the instance
```

```
cfddfc login -key ***** #allows to log into a cluster via ssh
```

```
mpirun -np 4 -hostfile $HOME/OpenFOAM/machines simpleFoam -parallel > log.simpleFoam & #runs calculations on 4 processors.
```

Note: it might be necessary to split domain into sectors prior to using this command eg: via the *decomposePar* command.

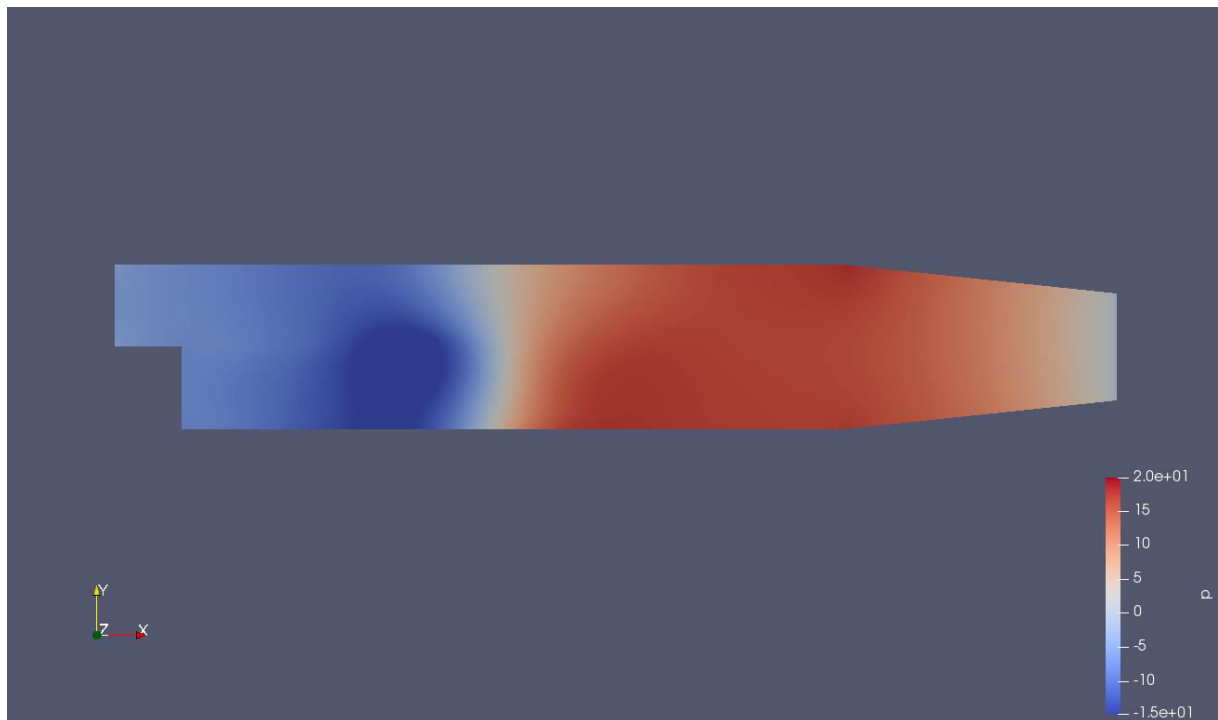
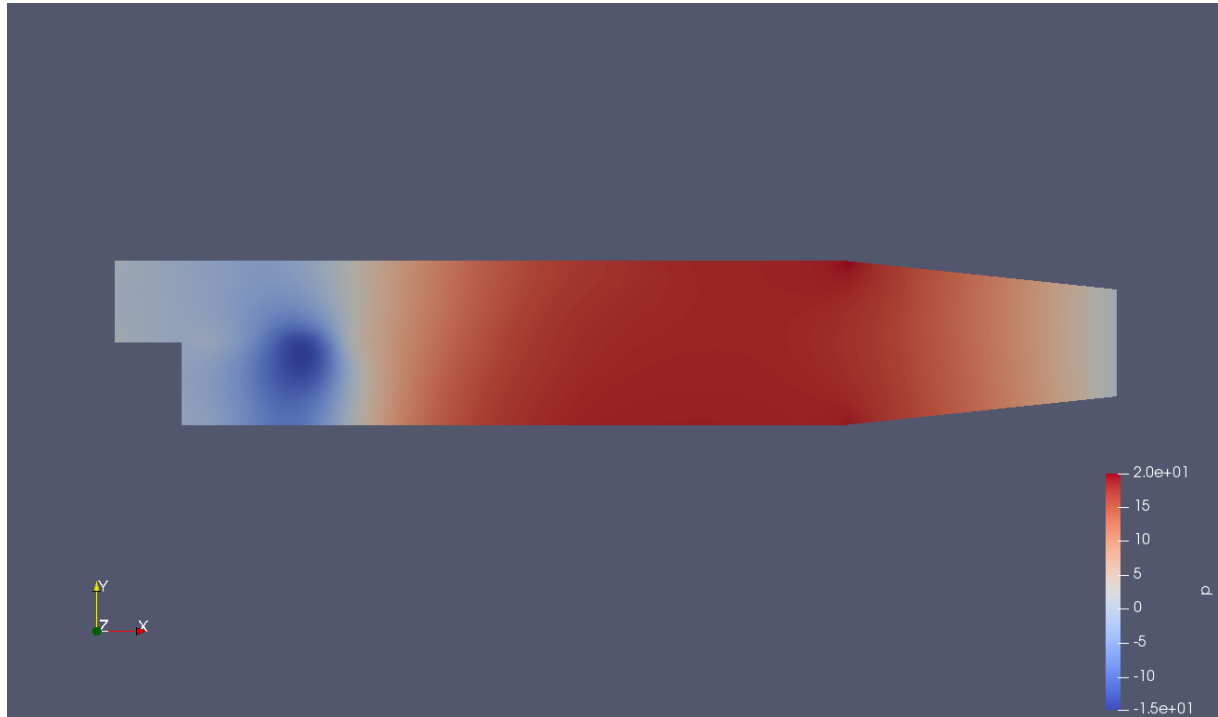
Results were then „reconnected” into one using a command *reconstructPar*.

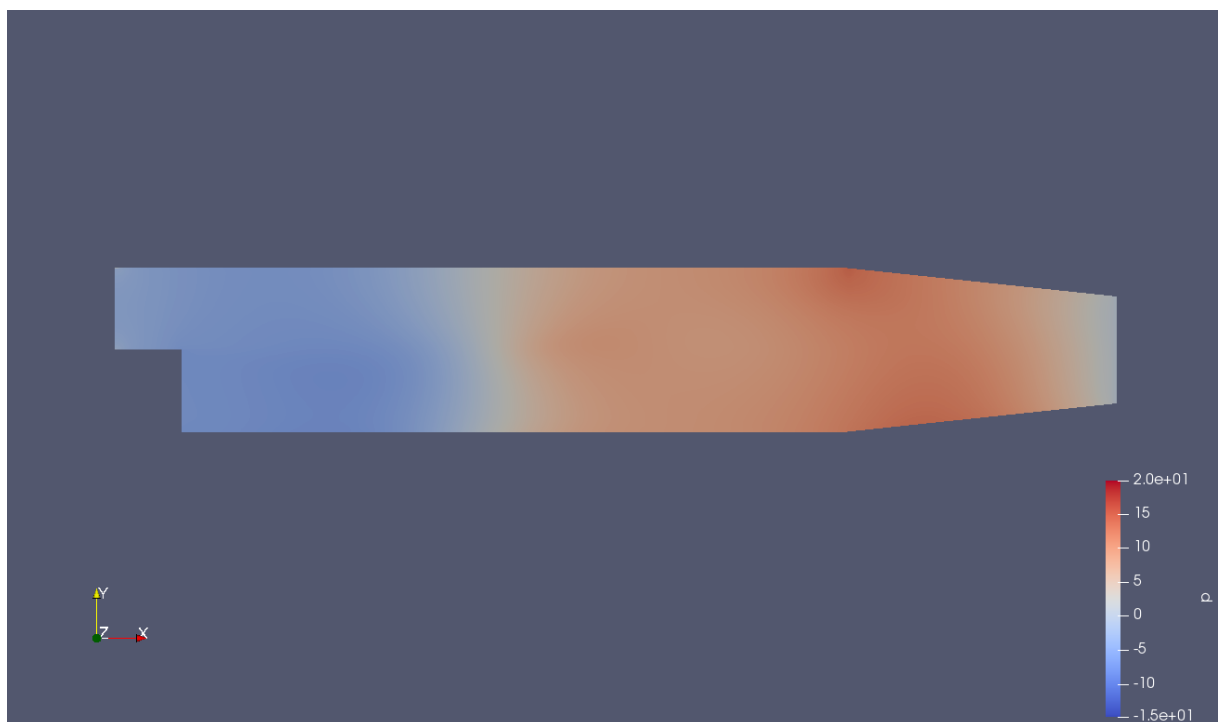
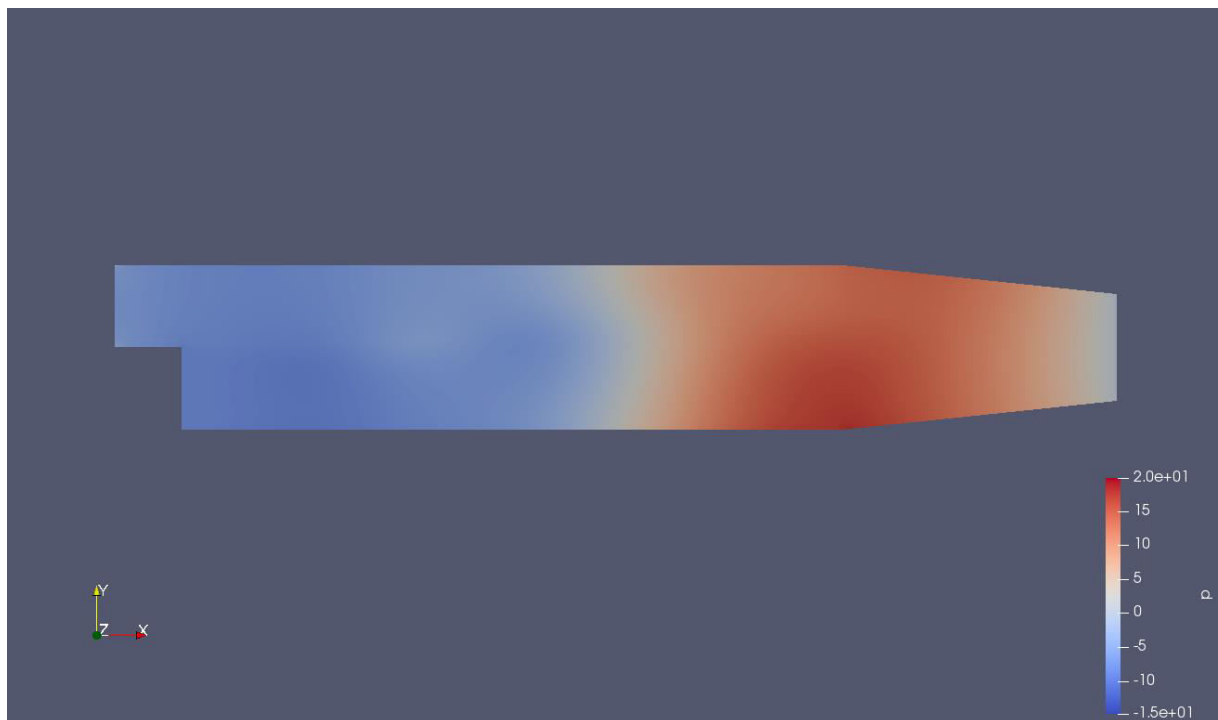
Furhtermore, in order to put the computational power of the cluster to a better use, the mesh was refined using command *refineMesh -overwrite*.

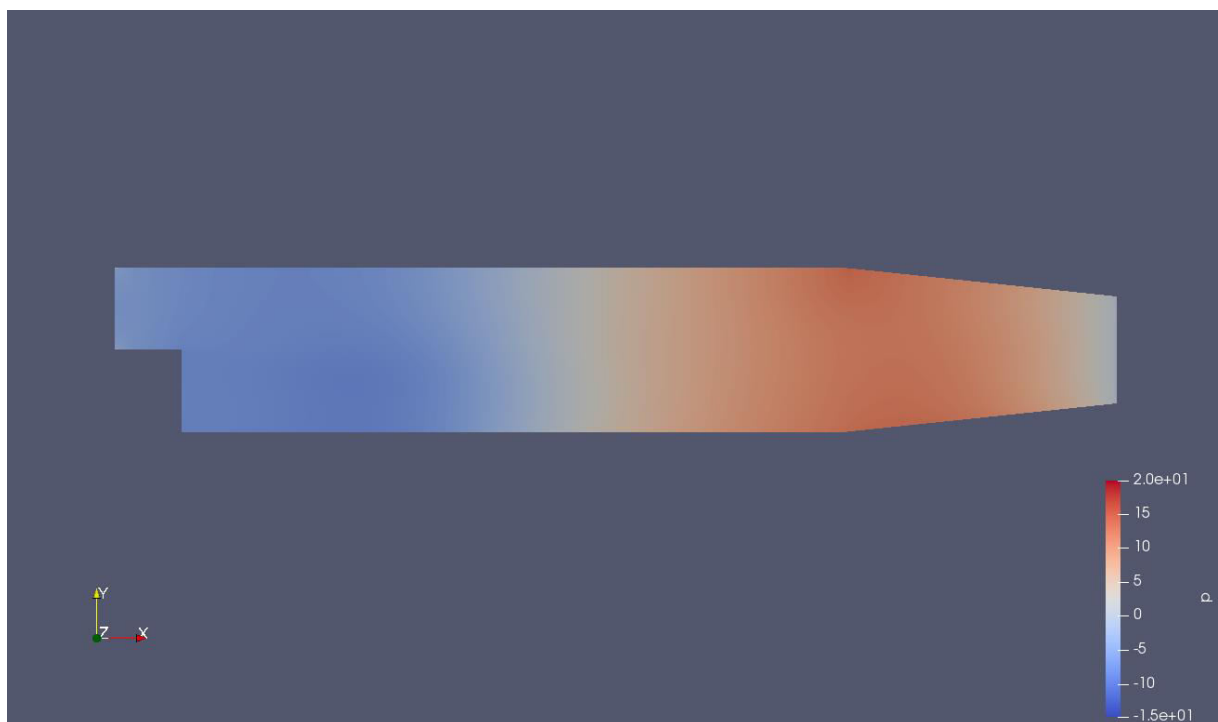
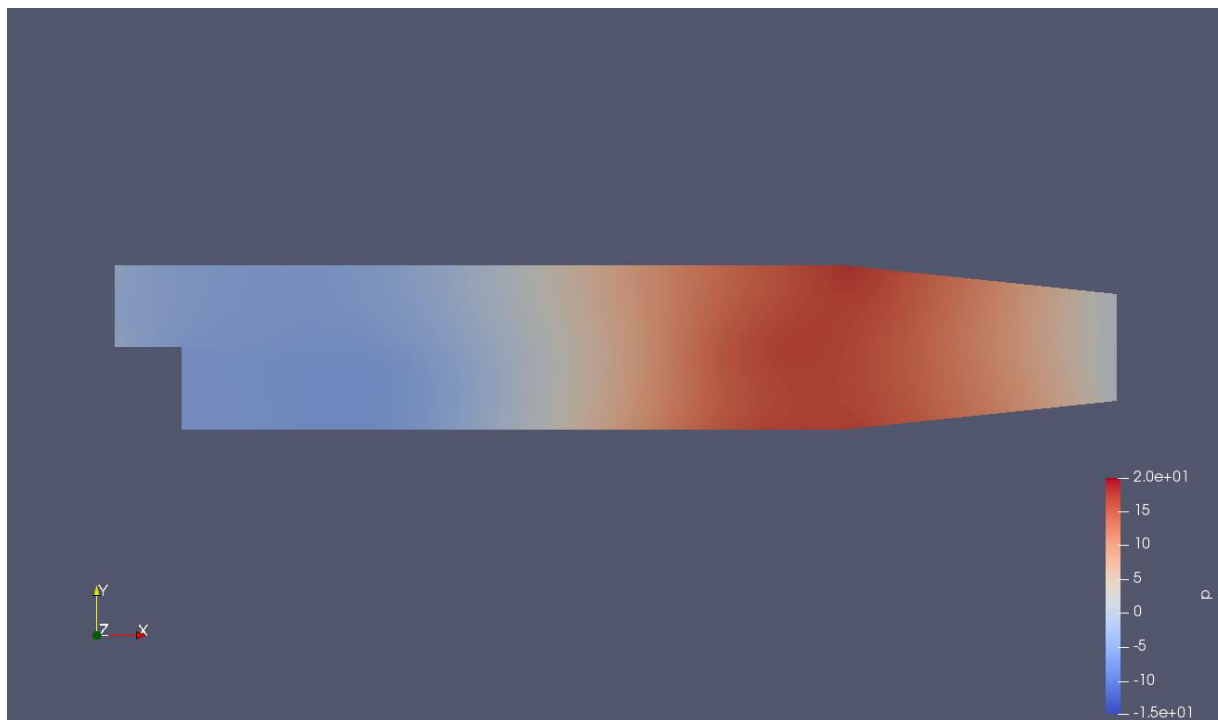
### 3. Results

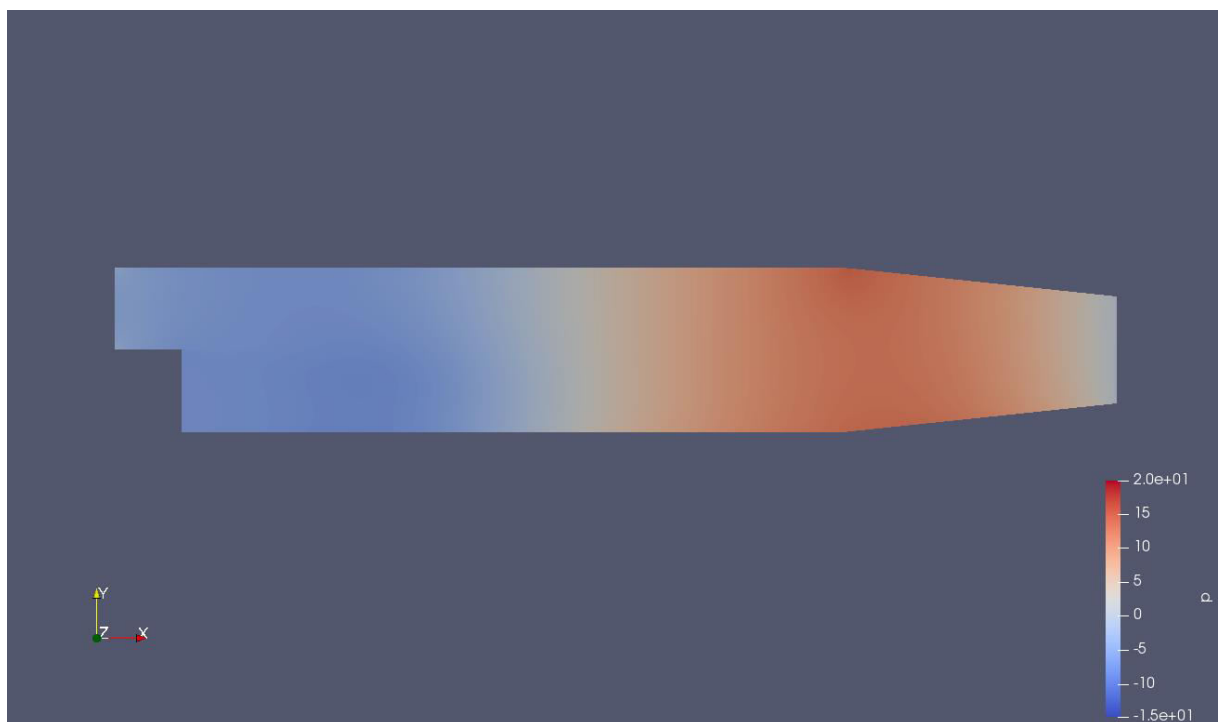
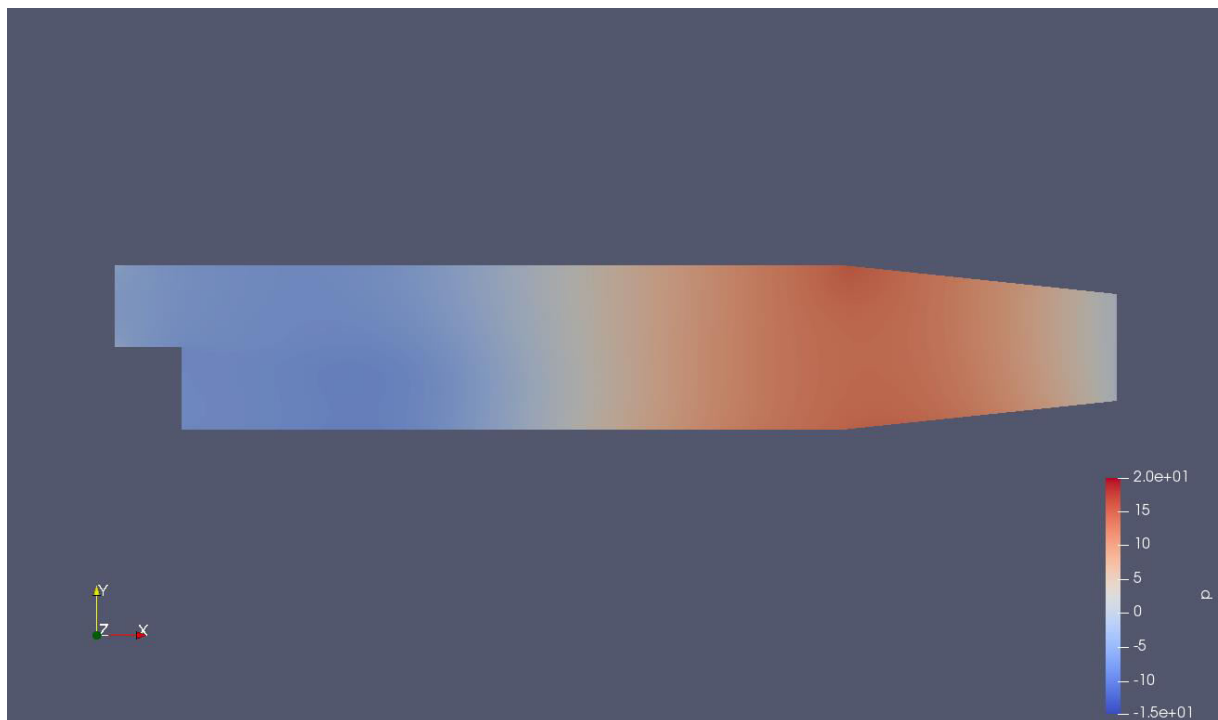
The results of this simulation are the pressure and velocity fields in various moments visualised using ParaView program. After some time we can observe that the flow becomes steady.

Pressure fields:

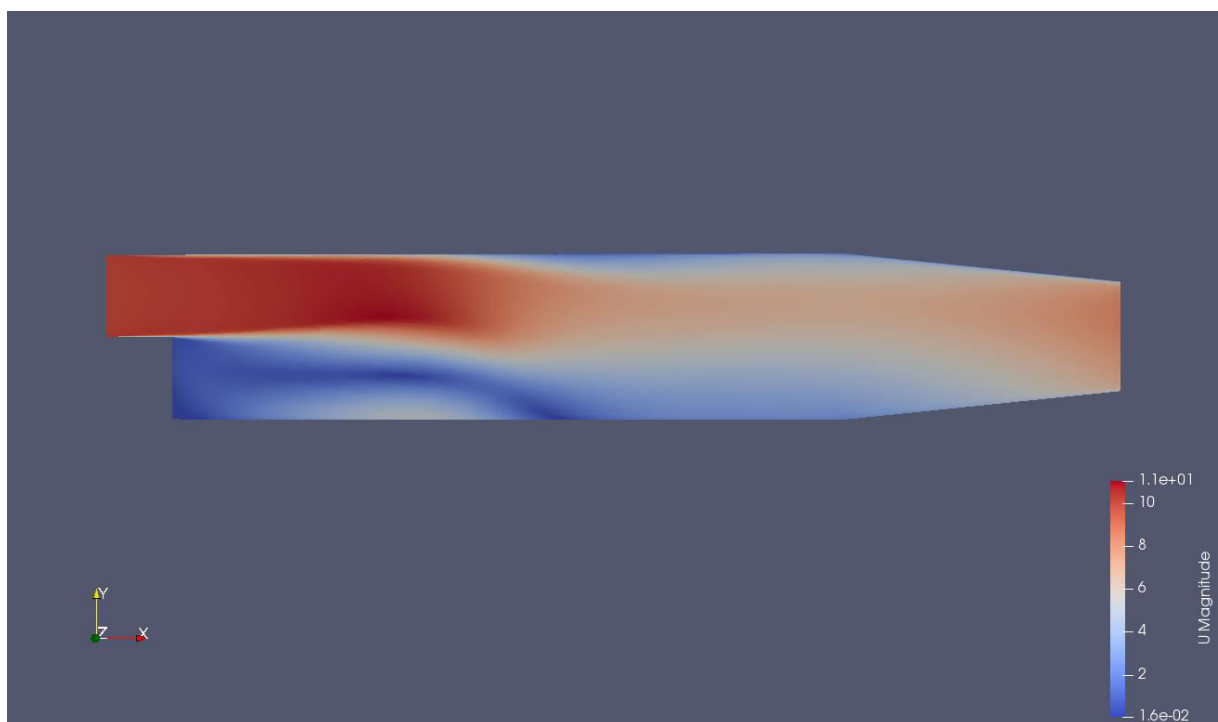
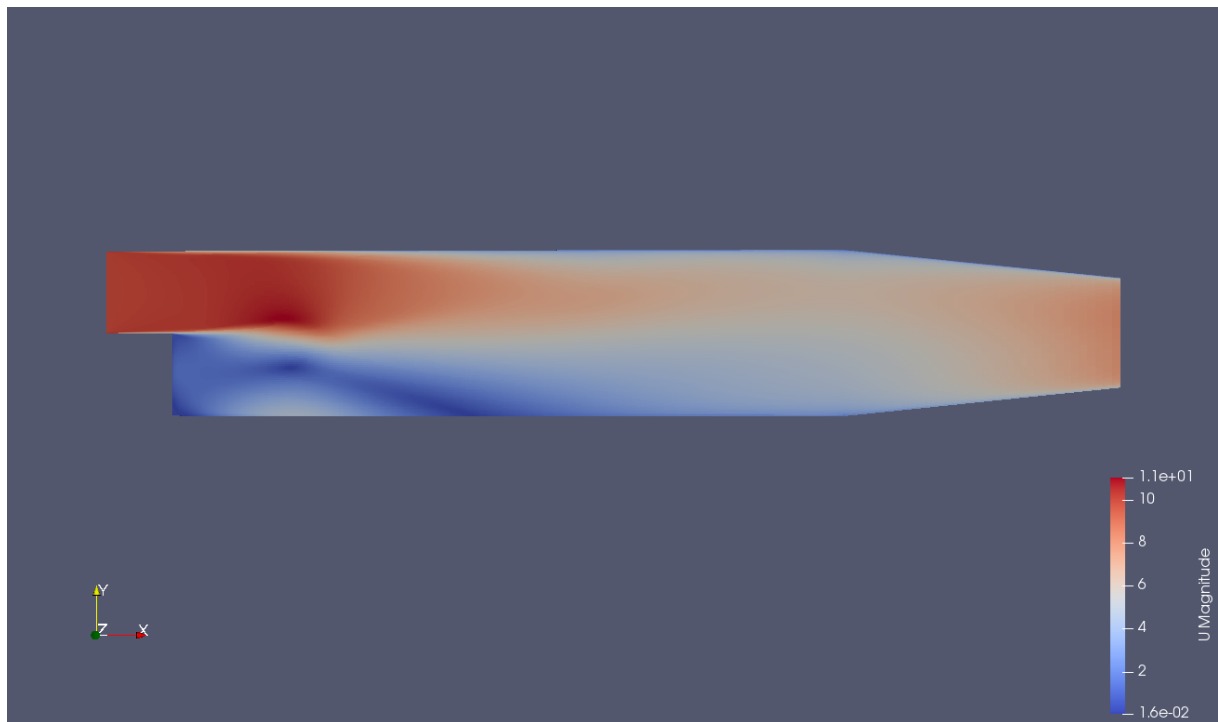




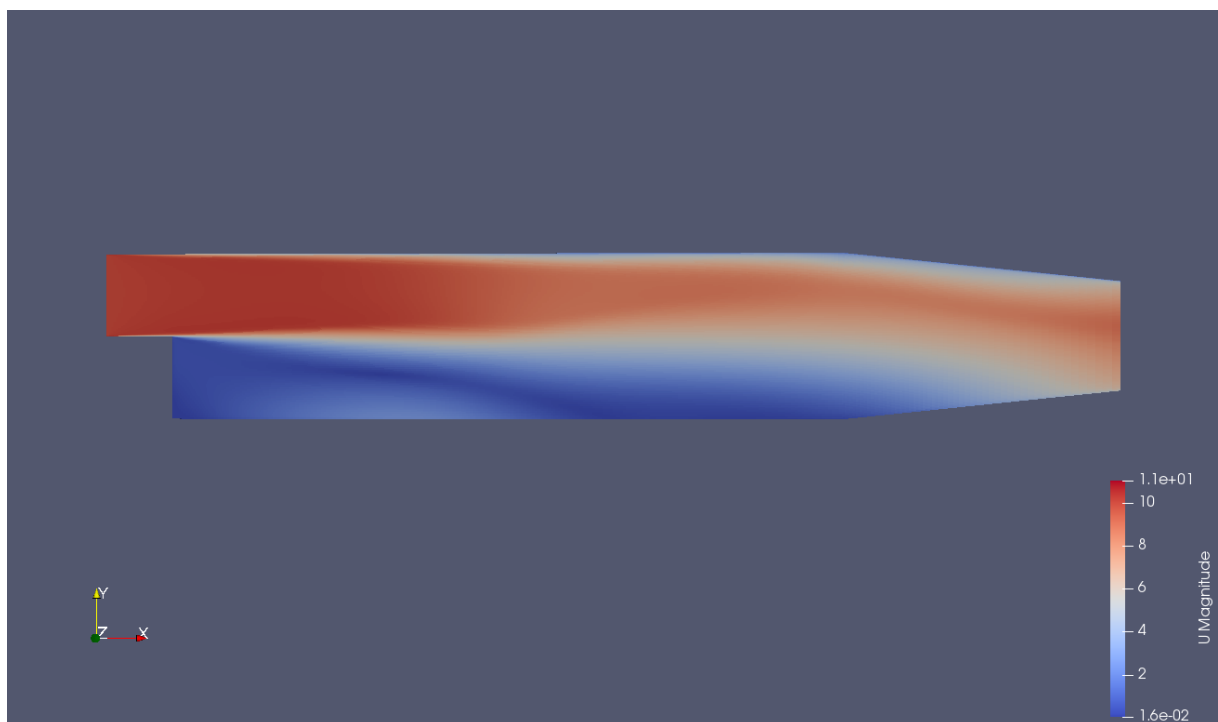
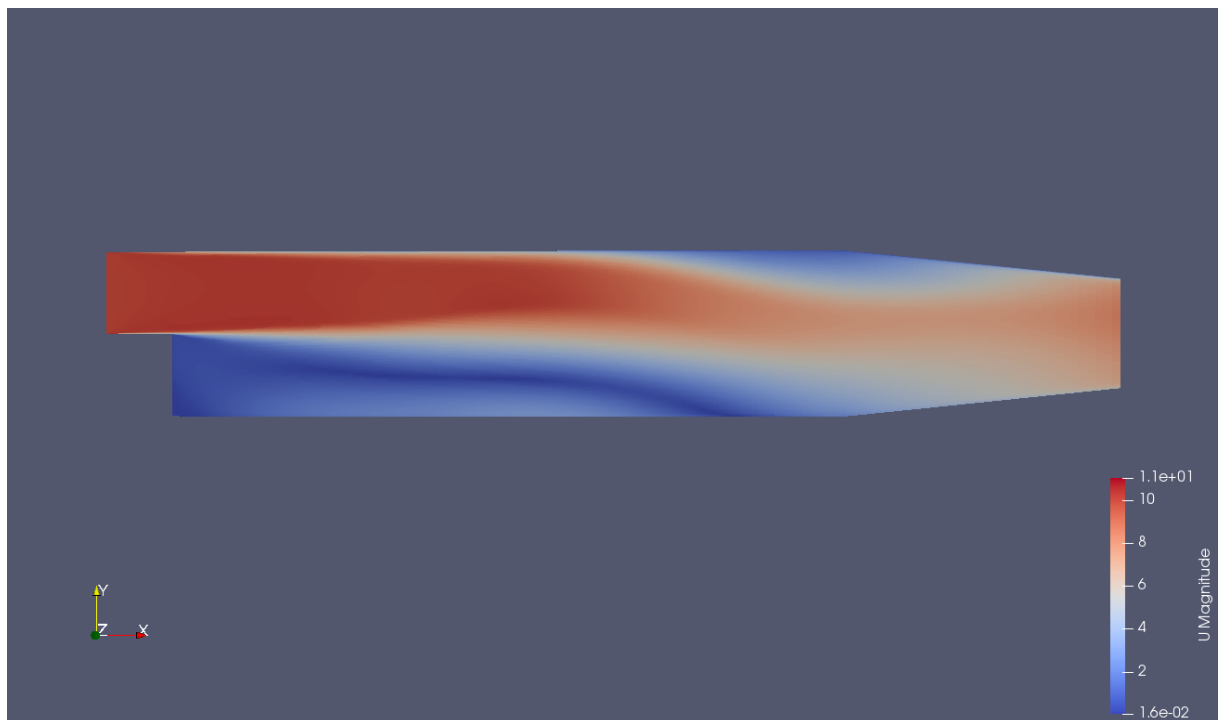


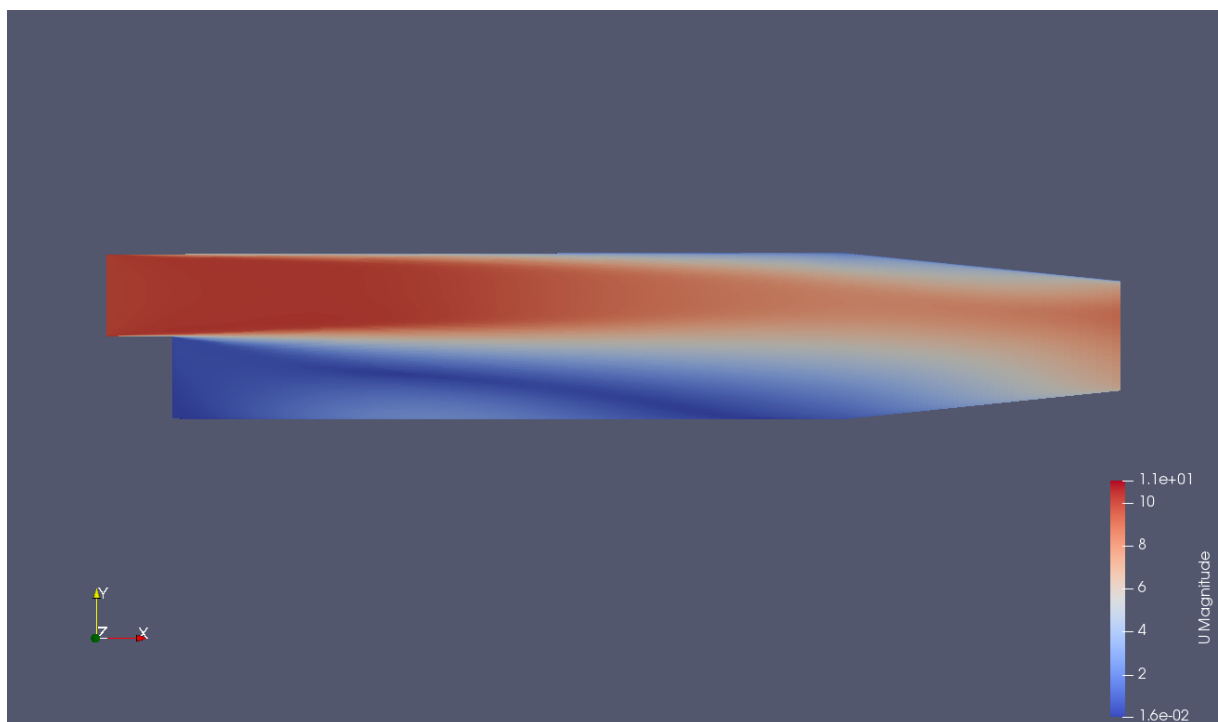
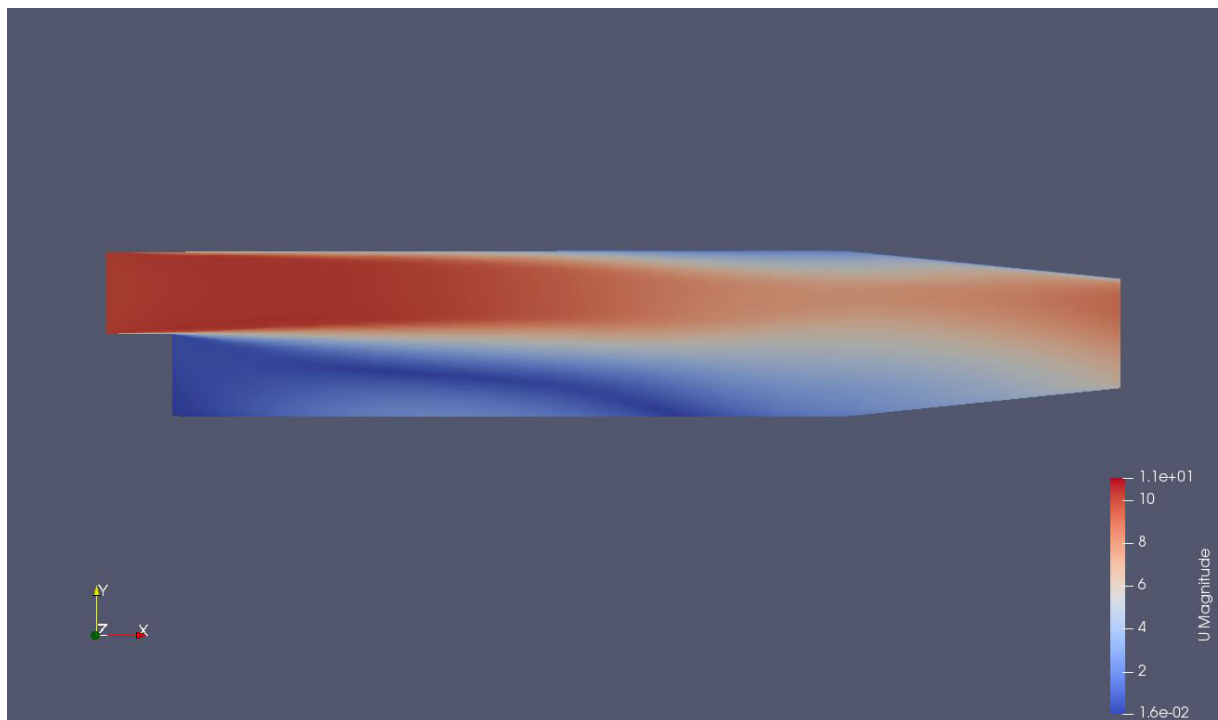


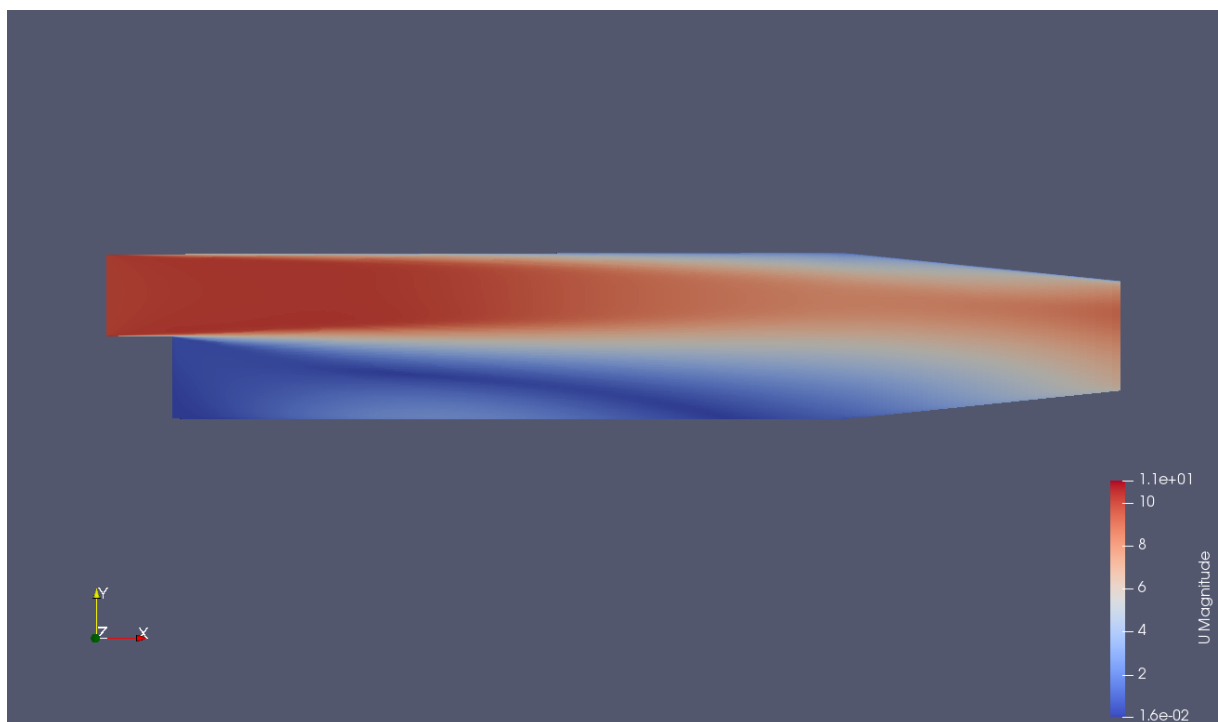
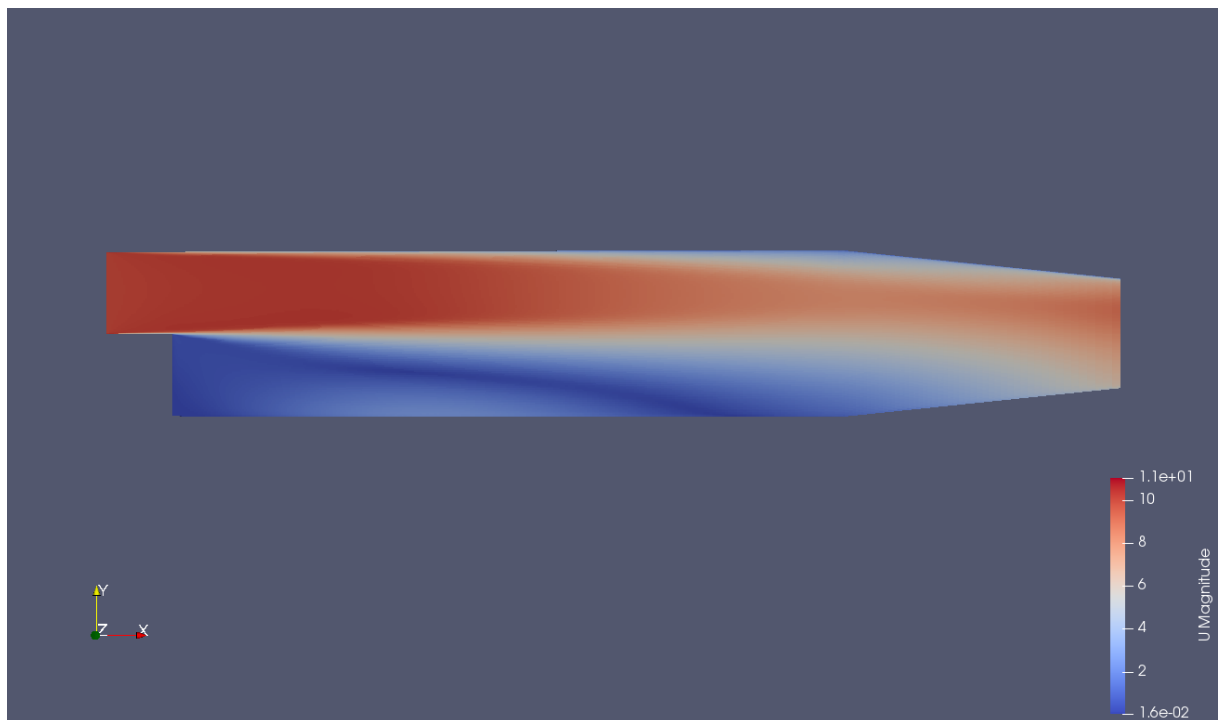
Velocity fields:











## 4. Conclusion

The use of Amazon Web Services proves to be a convenient, reasonably cheap and fast way to perform complex calculations of CFD cases. This method's greatest advantages are its availability, flexibility, easy setup as well as ability to be operated by low powered hardware from anywhere in the world with Internet access.

## 5. Sources

1. <https://cfd.direct/cloud/aws/>
2. [https://github.com/openfoamtutorials/openfoam\\_tutorials/tree/master/Parallel](https://github.com/openfoamtutorials/openfoam_tutorials/tree/master/Parallel)
3. <https://www.howtoinstall.me/ubuntu/18-04/paraview/>