Cloud Computing

Krzysztof Chodara

January 18, 2018

1 Introduction

Nowadays many computer applications require more and more computing power. Easy and economical way to increase computing power can be using cloud computing services. In this case AWS amazon service was used to compute CFD cases.

2 Obtaining Computing Instance

2.1 Configuring AWS account

Project was started by creating AWS account. Later these components were configured:

- IAM - security and access module
- S3 - storage module
- EC2 - computing power

Scope of this project was focused on computing so most of time was spent on EC2 configuring where user is capable to create computing instances from wide selection of them so there is possibility to match instance to computing power needs and user's funds. Linux instance was created which was free of charge. Result was connection addresses and private key.

2.2 Configuring Linux Instance

To configure the instance putty application on client computer with windows was used. To login IP address and private key was used. Then python interpreter and additional libraries were installed. Python scripts were transfered by Winscp application. Unfortunately adding matplotlib library ended with failure.

To get around that problem Jupyter was installed, on Linux instance, and started. Attempts to connect it thought ssh connection between user's computer with windows and server with Linux ended with failure so virtual machine was configured on user's computer and Ubuntu was installed. Then connection between user's computer (via Ubuntu) and Jupyter instance was established.

So the Linux instance, on AWS server, was ready to run this project's scripts.

3 CFD Scripts

3.1 Preparing Navier-Stokes Script

To prepare N-S equation script, prof. Lorena A. Barba tutorial was used. Some extra steps were added for example: disabling false positive errors, repairing plots and adding extra plot. Script present flow between two walls. See fig.1-2.

3.2 Von Karman Vortex Calculations

For calculating flow over cylinder, in 2D, open-source code was used. Minor changes were applied to code. See fig. 3-4

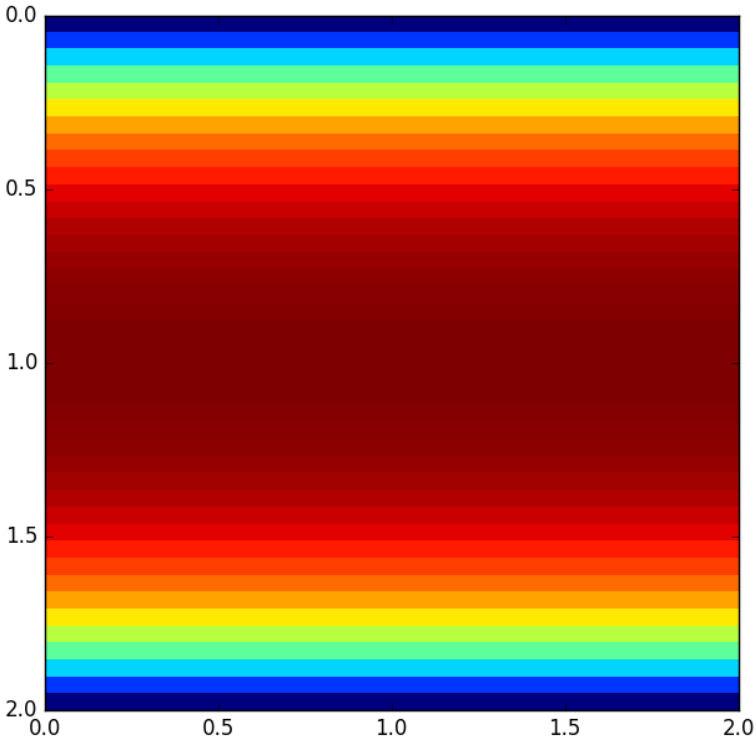

Figure 1: Velocity contour

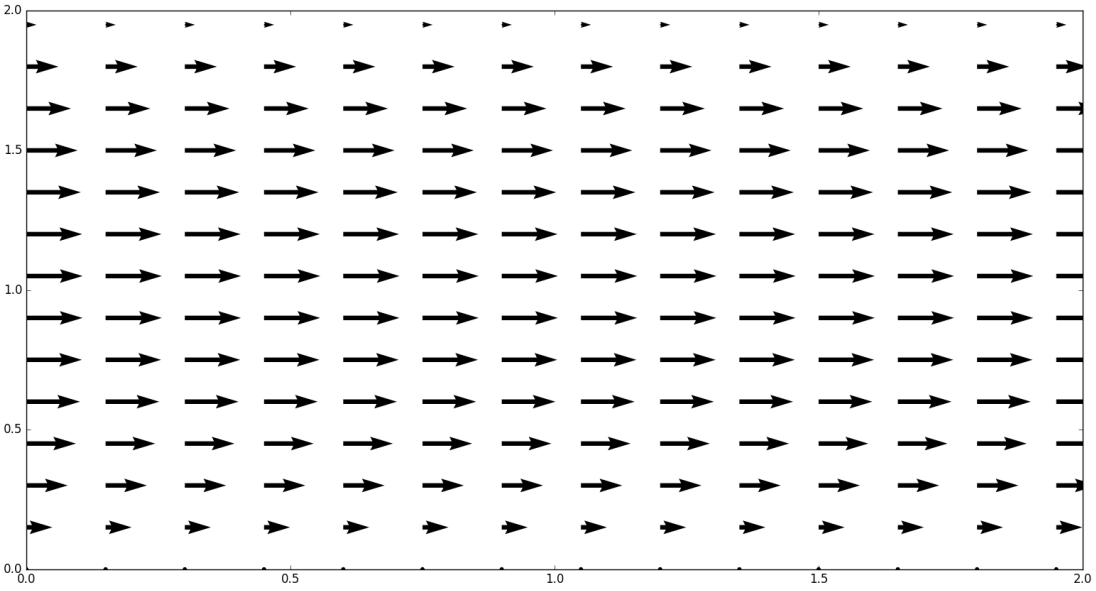

Figure 2: Velocity vectors

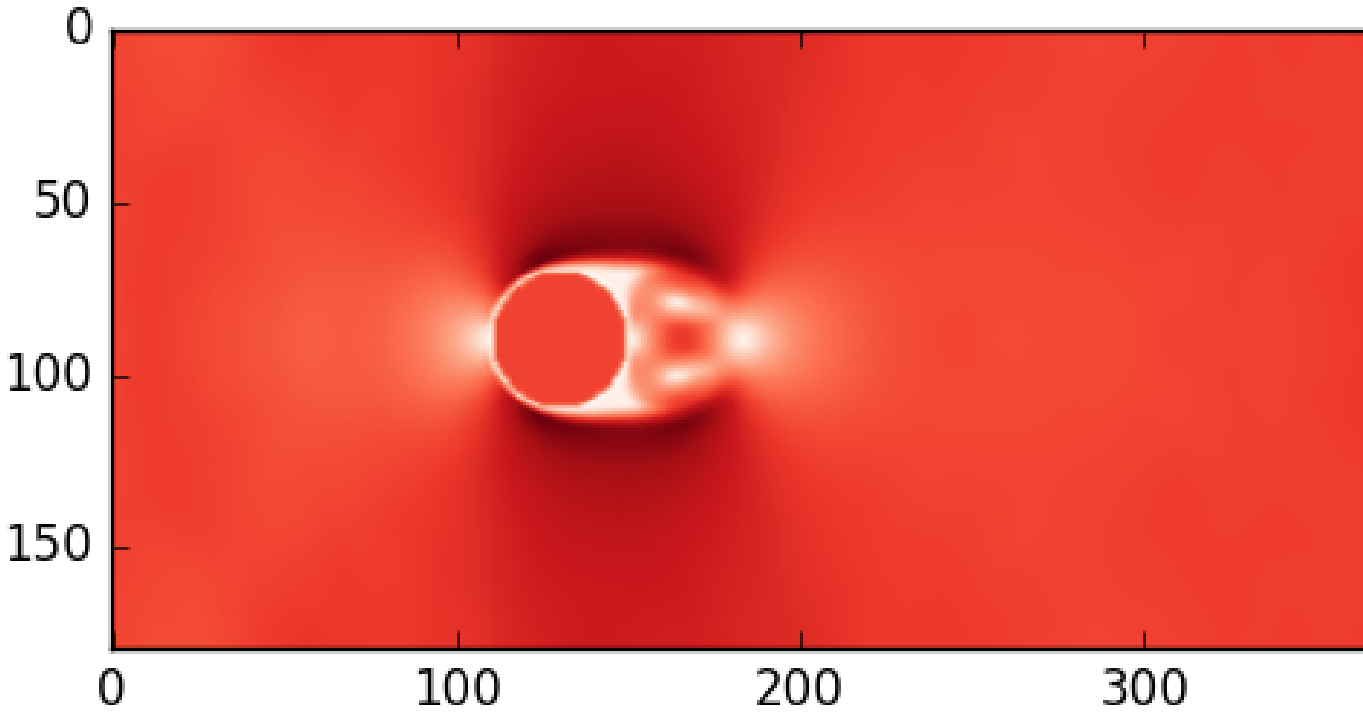

Figure 3: Flow at beginning of simulation...

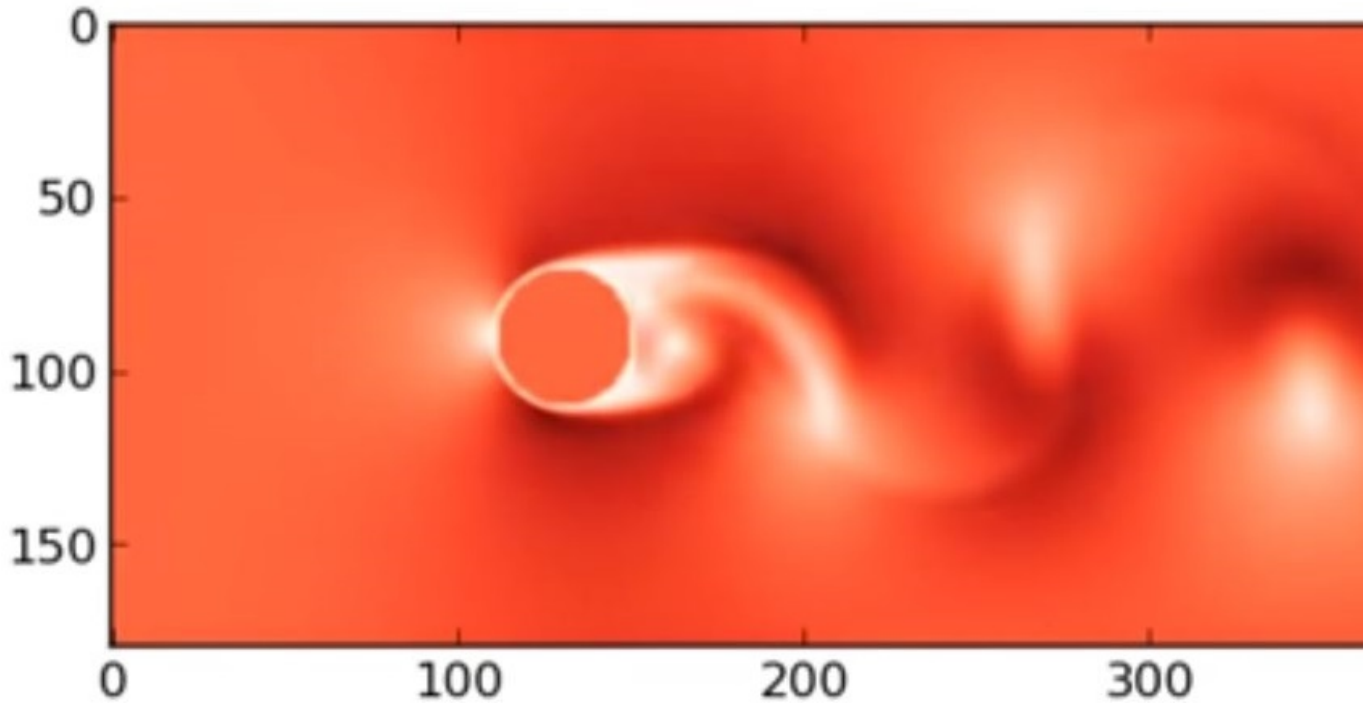

Figure 4: ... and at the end

4 References

- https://docs.aws.amazon.com/AWSEC2/latest/UserGuide/AccessingInstancesLinux.html
- http://lorenabarba.com/blog/cfd-python-12-steps-to-navier-stokes/
- http://wiki.palabos.org/