
Symulacje Komputerowe Procesów Spalania

Aerodynamical Analysis of NACA 0012 Airfoil

Author: Michał Górski

Lecturer: dr inż. Mateusz Żbikowski

Warsaw University of Technology
Faculty of Power and Aeronautical Engineering
Warsaw, August 2018

Contents

Introduction	1
1 Preprocessing	1
1.1 Geometry	1
1.2 Mesh	2
1.3 Boundary conditions and models	2
1.4 OpenFOAM case setup	3
2 Results	4
2.1 Velocity contours	4
2.2 Pressure contours	5
2.3 Lift coefficient	6
3 Conclusion and summary	6
References	7

Introduction

The purpose of this project is to demonstrate capabilities of open-source CFD software OpenFOAM by using it to analyse two-dimensional flow around the popular NACA 0012 airfoil, and validate obtained results using ANSYS Fluent.

1 Preprocessing

1.1 Geometry

The geometry of computational domain was created in *ANSYS Design Modeler* by subtracting a surface created from coordinate points defining contour of mentioned airfoil from a C-shaped surface acting as a bounding box.

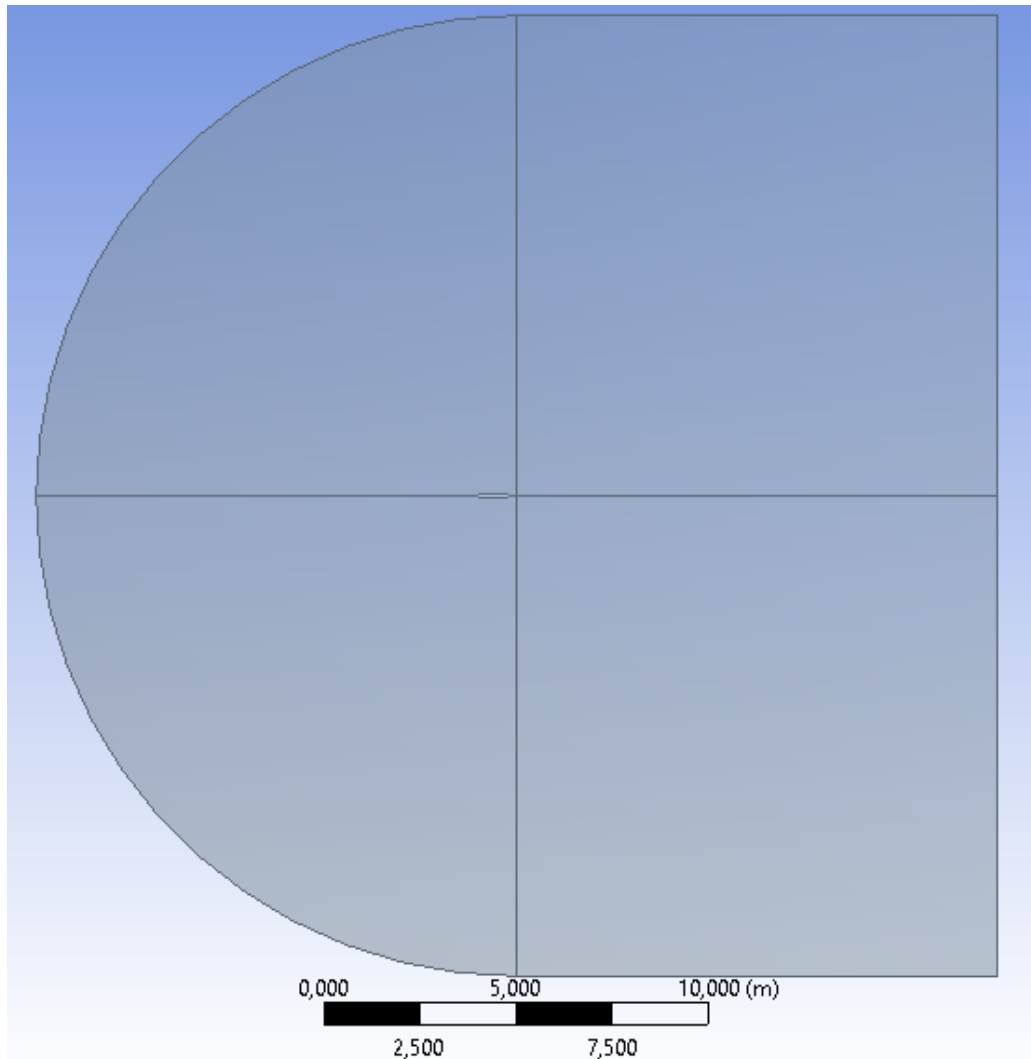
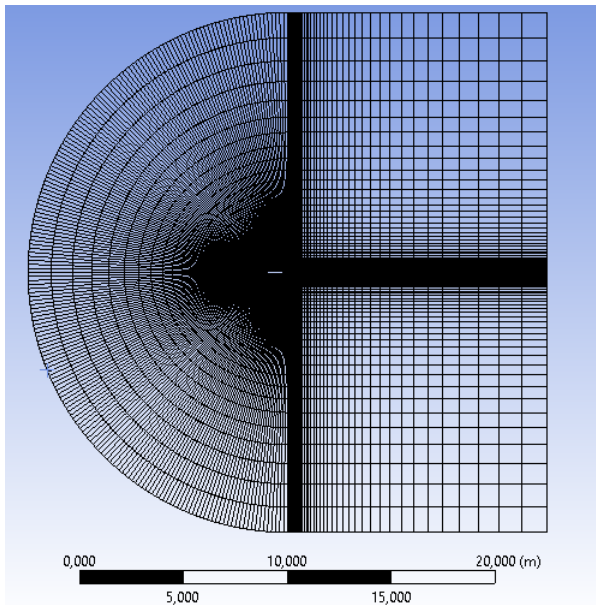


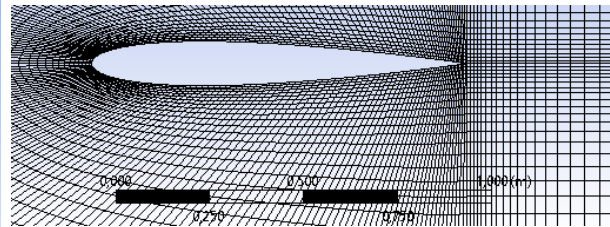
Figure 1: *Geometry of the computational domain.*

1.2 Mesh

Mesh of the computational domain was created in *ANSYS Mesher*. As it can be seen on the pictures below, the mesh is significantly more dense in vicinity of the airfoil in order to capture significant changes which pressure and velocity of the flowing air are prone to. Worth mentioning is the fact the mesh had to be exported in ASCII format which is the format of choice for OpenFOAM - ANSYS works on binary, therefore it is default and had to be change in Tools → Export prior to export.



(a) Mesh of the whole domain



(b) Closer look at the airfoil region

Created mesh contained 15000 elements and 15300 nodes.

1.3 Boundary conditions and models

The airfoil was of type wall, whereas the C-shaped line along with two bounding horizontal lines acted as an inlet. The vertical line, bounding the domain from the right-hand side was defined as an outlet.

Inlet:

- velocity magnitude $1 \frac{m}{s}$, vector components $[0.9945 \ 0.1045 \ 0]$
- velocity vector components defined in order to simulate an angle of attack of about 6°
- turbulent viscosity ratio set to 10
- pressure 0 Pa

Outlet:

- velocity magnitude set exactly the same way as in the inlet - computational domain is large compared to the size of the airfoil, it was easier to leave it like that in the file
- turbulence viscosity ratio set to 10
- pressure 0 Pa

Wall:

- stationary, noSlip and zeroGradient type

Turbulence model was set to Spalart-Allmaras, rho parameter for the fluid was set to the value of 1, and kinematic viscosity to 1e-5. All equations were discretized using first-order schemes.

1.4 OpenFOAM case setup

To perform this analysis, tutorial *airfoil2D* was adapted. ANSYS-made mesh was converted to OpenFOAM format using the following command:

```
fluentMeshToFoam -2D 1 -writeSets -writeZones airfoil.msh
```

where `airfoil.msh` is the name of the file containing mesh that was used for this analysis' purposes.

Another difference between this case and *airfoil2D* tutorial case, aside from different boundary conditions and turbulence model, was the presence of `forceCoeffs` file. This file allows the user to read forces, force coefficients and moments acting on the model. These parameters are of extreme importance in regard to airfoils.

The `forceCoeffs` file has to be created in case's `system` directory, and following lines of code have to be typed into the file `system/controlDict`:

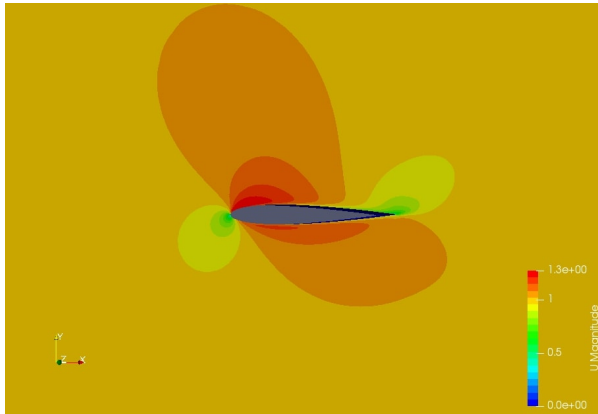
```
functions
{
    #include "forceCoeffs"
}
```

Calculations were carried out in 3000 iterations, 1 second each, which was more than enough to stabilize forces and residuals.

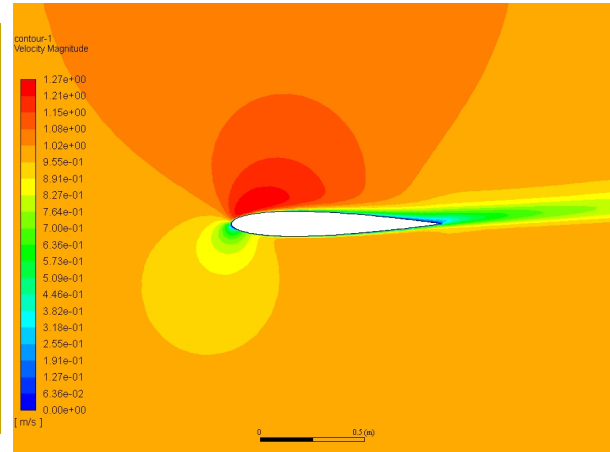
2 Results

2.1 Velocity contours

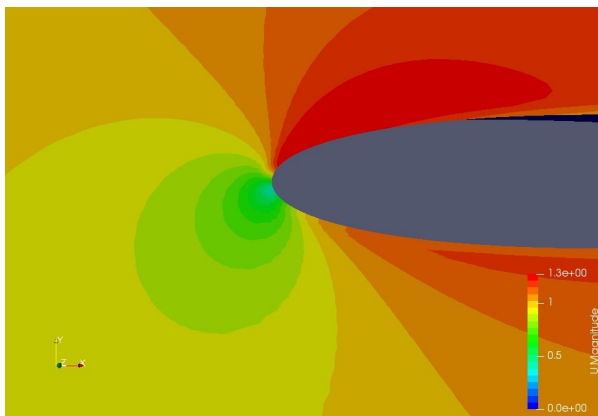
Following pictures present velocity contours around the airfoil obtained both using OpenFOAM simpleFoam solver (left) as well as ANSYS Fluent (right). Scales are the same.



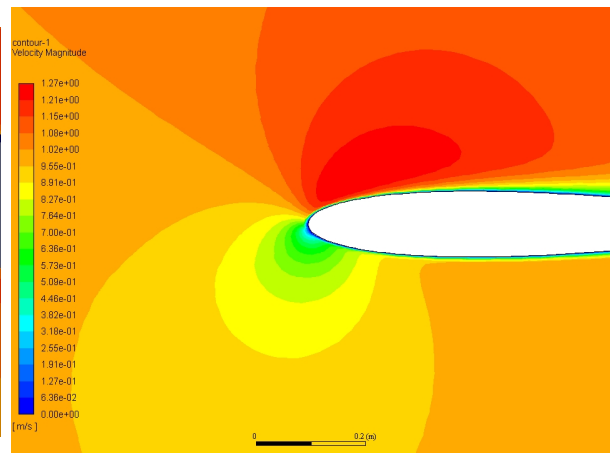
(a) OpenFOAM velocity contours



(b) Fluent velocity contours



(c) OpenFOAM velocity contours

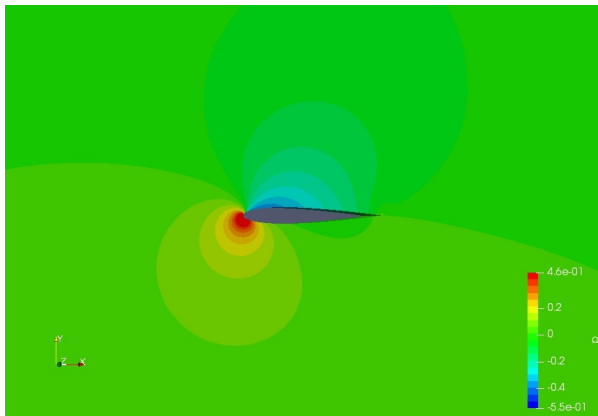


(d) Fluent velocity contours

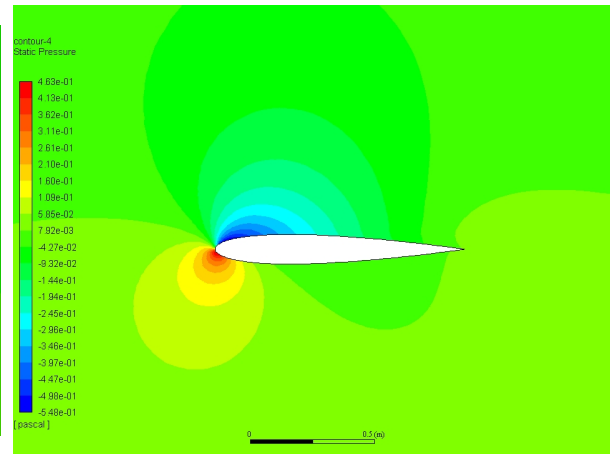
The fluid can be observed being sped up when negotiating the bend on top of the airfoil's leading edge. According to Bernoulli's Principle, it should result in pressure drop on the top side of the airfoil, which contributes to lift generation.

A significant difference between shown velocity maps can be observed, fluid in OpenFOAM case acting like it was "glued" to the airfoil. It may indicate erroneous fluid parameter input.

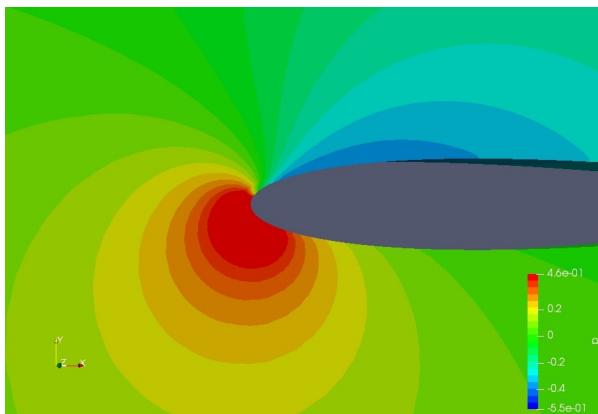
2.2 Pressure contours



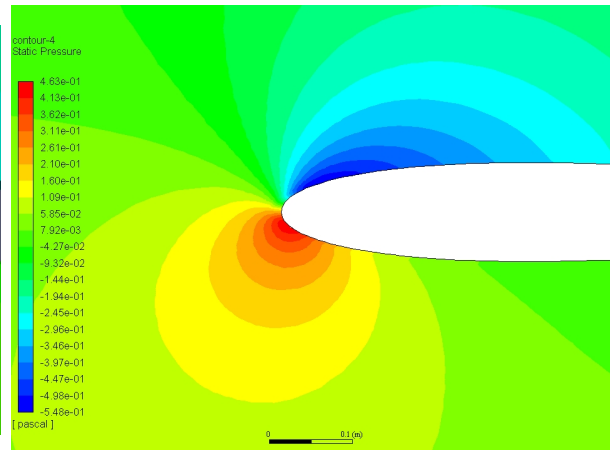
(a) OpenFOAM pressure contours



(b) Fluent pressure contours



(c) OpenFOAM pressure contours



(d) Fluent pressure contours

Pressure contours shown above are consistent with what could have been concluded from velocity maps - fluid's velocity increases over the airfoil's top part and it results in pressure drop. Since pressure over the airfoil is lower compared to pressure under the airfoil, lift force is generated.

2.3 Lift coefficient

Lift coefficient obtained from OpenFOAM is equal to 0.625, which is 25.5% higher than the coefficient calculated by Fluent - 0.498. It may result from previously mentioned erroneous input of fluid parameters.

3 Conclusion and summary

OpenFOAM is a free CFD toolbox, which offers the user great amount of control over the flow of the program, what cannot be said about expensive commercial software like ANSYS Fluent. On the other hand, OpenFOAM is much harder to use for an average user with no significant programming background, and its solvers' robustness is lower compared to Fluent.

OpenFOAM 6 was used to perform described analysis along with ParaView 5.4 for postprocessing. OpenFOAM was installed on Ubuntu set up in Windows Subsystem for Linux environment, available on Windows 10 since the Fall Creators Update.

References

- [1] <https://confluence.cornell.edu/display/SIMULATION/Flow+over+an+Airfoil+-+Mesh>
- [2] <https://www.hpc.ntnu.no/display/hpc/OpenFOAM+-+Airfoil+Calculations>
- [3] <https://cfd.direct/openfoam/user-guide/v6-turbulence/>