KEB-45250 Numerical Techniques for Process Modeling
CFD Assignment

Antti Mikkonen

March 10, 2018

Contents

1   Introduction

This course contains three assignments: two about CFD and one that will be published later. At least one of the CFD assignments and the assignment that will be published later must be passed. It is also possible to return all three assignments. This makes it possible to earn 5-10% extra points. See Sec. 4 for details.

The first of the CFD assignments is a classical 2D NACA 0012 airfoil turbulence model validation case. It is assumed that this assignment is completed with Fluent or OpenFOAM. If you wish to use some other tool, ask first. See Part I for details.

The second CFD option is a 2D Blasius boundary layer solution with finite volume method using a custom code. In this case, the pressure term and many other terms in the general solution can be ignored. This makes the solution much easier. It is assumed that the code is written in Python. If you wish to use some other language, ask first. See Part II for details.

2   Submitting the report

Both CFD assignments will be reported and the report submitted to antti.mikkonen@tut.fi in pdf format.

Deadline Sunday 29.04. at 23:55.

I will send an answer email withing 24 hours. If you receive no such email, assume that something went wrong and contact me.

Both of the CFD assignments are classics and it should be easy to find references. This in encouraged. It is, however, required that you complete the calculations by yourself.

If you use some third party sources, cite them in a good scientific manner. If you are unsure about the ethics of using something, ask. If suspicion arises, I will arrange an oral exam. Do not lose your Fluent etc. files.

The report may be written in Finnish or English.

The report must be written in good scientific style with citations at appropriate places. Equations, figures, etc. must be numbered and referenced in the text.

The subject matter is the main component for grading. For high grade, however, good style is also required.

Aim for compact, clear presentation. The equations and figures are likely to contain the most important information.

3   Scientific writing tools

Your report is likely to contain many long equations and a lot of figures. Keeping track of the references manually becomes very tedious, very fast. I would highly recommend using some tool designed for scientific writing. Usually, that means some LaTeX variant.

My personal favorite is LyX (https://www.lyx.org/Screenshots). It combines the automated functionality of LaTeX to a graphical user interface. Especially equation typing is much easier in a little more graphical environment. Equation number, figure numbers, citations, cross-references, etc. are handled automatically.

This pdf has been written in LyX and contains most of the elements you will need in your report. The raw LyX file is attached. If you are new to LyX, I would recommend using this file as a template. I am happy to help with the report also.

You may, of course, produce your report any way you like.

Fake citations for demonstration [2], [1, 3]

4   Course grading

The exact method for weighting the grades will be decided later.

- Exam
    - must be passed, grading 0-5
    - 60% weight

- CFD assignments
    - one must be passed, grading 0-5
    - 25% weight
    - If you do both, you can get 5-10% extra
    - The report must be completed in time
        * Late, acceptable return results in grade 1
    - Returned report may be
        * accepted
            · grading 1-5
        * accepted with corrections
            · if it seems obvious that the student has understood the subject, but made some simple mistake
            · after corrections, grading 0-5
        * failed with corrections
            · large errors
            · after corrections, grade 0-1
        * failed
            · grade 0

- The assignment that will be published later
    - must be passed, grading 0-5, 15% weight

Part I
NACA 0012

5   Introduction

We use the classic NACA 0012 airfoil, see Fig. 1, validation case for our assignment. See NASA for details (https://turbmodels.larc.nasa.gov/naca0012_val.html). Note that the CFD studies references on the NASA page are quite old. Modern models are easier to use.

It should be easy to find newer references as well. It is recommended to spend a while learning how large a domain, how fine a mesh, and what kind of turbulence model you need. The case can be solved in 2D and steady state.
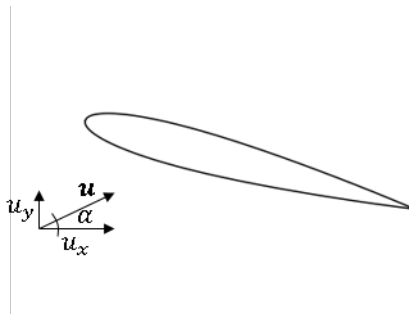

Figure 1: NACA 0012

The measured values are usually given in dimensionless units. It should not matter how you choose the dimensions, but the values in Table 1 work.

6   Input values

The angle of attach is based on the last digit of student number. All other parameters are the same for everyone, see Tables 1-2. If care is taken with the dimensionless number, other dimensional values can also be used.

	Value	Name
α	See Table 2	angle of attach
Re	6×10^6	Reynolds number
μ	$1.02 \times 10^{-5}\, Ns/m2$	dynamic viscosity
ρ	$1.03\, kg/m^3$	density
L	$1\, m$	chord length

Table 1: Inputs

last digit	1	2	3	4	5	6	7	8	9	0
α (°)	3	4	5	6	7	8	9	10	11	12

Table 2: Choose the angle of attach based on your student numbers last digit. Example: 123456 -> $\alpha = 8\,°$

7 Provided files

- Measurement data from NASA

  – filesForNACA0012/LadsonForceData.txt

- This pdf, and the LyX files used to create it

  – filesForNACA0012/lyxRawFiles

- Geometry files for structured mesh and unstructured meshes

  – filesForNACA0012/geometry

- An intentionally coarse and small mesh for testing. You need to make you own meshes.

  – filesForNACA0012/mesh

- Some code that might be useful

  – filesForNACA0012/code_snippets.py

- Chosen tutorials, provided by EDR&Medeso

  – filesForNACA0012/tutorials

8 Include in your report

- Calculate drag and lift coefficients

  – In this case, CFD should give very food results
  – On the NASA page, see the What to Expect section
  – For example, SST model gives very good results (https://turbmodels.larc.nasa.gov/naca0012_val_sst.html)

- Two turbulence models

  – Discuss the differences and similarities between models
  – It is required for one of the models to give comparable results to those reported by NASA

- Mesh Independence study

- – You may use a mesh from a 3rd party for comparison
  - – table would be a good format
  - – at least one figure of a mesh
    - * you may use a coarse mesh for visualization
- Mesh quality
- Sensitivity study for domain size
  - – table would be a good format
  - – Increase domain size in steps
  - – once the solution do not change, the domain is large enough
- Discretization schemes
- Convergence
  - – Monitors of lift and drag are a good criteria
- Governing equations
  - – For turbulence models, the transport equations and turbulent viscosity definition are enough
- Boundary conditions
  - – also for walls
  - – Some wall functions are very complicated. Just explain the basics. If you are using Fluent, the Fluent theory guide is a good source.
- Qualitative flow field visualization
  - – Velocity field and pressure field contours, perhaps
  - – Velocity vectors or streamlines
- Only discuss one simulation in detail
  - – There is no need to plot results of the cases used in the mesh independency study, for example
  - – Use tables or charts to compactly explain than you have done tried other meshes, domain sizes, etc.

9 Bag of tricks

- Your calculations are likely to be sensitive to

  - near wall mesh quality
  - near wall turbulence
  - size of the domain. You will likely need a large domain

- You can use correlations to estimate how fine near wall mesh you need. One such correlation is implemented here:

  https://www.cfd-online.com/Tools/yplus.php

- The Fluent Theory manual, in ANSYS help, is very useful

- Youtube has a lot of tutorials. They are usually made with older versions but the differences are minor.

- When using structured grids, large jumps in mesh size may cause discontinuities in the field values

- Similar, but not the same, tutorials

  - https://confluence.cornell.edu/display/SIMULATION/ FLUENT+-+Flow+over+an+Airfoil
  - WS04_Airfoil tutorial in the Fluent quick start by EDR&Medeso
  - 2D Combustion Chamber meshing tutorial by EDR&Medeso

Part II
Blasius boundary layer

10 Introduction

Blasius boundary layer is a well known classic of fluid dynamics. See, for example, Wikipedia (https://en.wikipedia.org/wiki/Blasius_boundary_layer) Blasius boundary layer has a well known analytical solution. In this assignment, the same case is solved using finite volume method and a custom code. The analytical solution is used for validation.

The assumed language is Python. If you wish to use some other language, ask before starting the assignment.

The simplified governing equations of the Blasius boundary layer are

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = \mu \frac{\partial^2 u}{\partial y^2} \tag{2}$$

Note that the equations given in Wikipedia have a pressure term. We ignore it. The analytical solution is explained, for example, in here https://www.calpoly.edu/~kshollen/ME347/Handouts/Blasius.pdf.
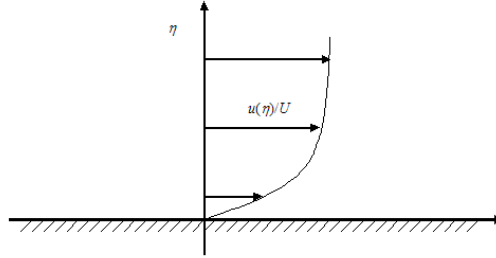

Figure 2: Blasius boundary layer

Because of the ignored pressure gradient, the Eqs. 1 and 2 are considerably easier to solve than more general Navier-Stokes equations. As this assignment is outside the scope of the exercises done on this course, it is likely it will require a lot of work.

Many other terms are also excluded during the derivation of Eqs. 1 and 2. You may include the excluded terms in your finite volume solution or exclude them.

The Blasius boundary layer is well known and it should be easy to find references. It is recommended to use them. Just remember to cite the sources and to write your own code.

There are some Python libraries for Finite Volume Method, for example FiPy (https://www.ctcms.nist.gov/fipy/). This assignment would be too easy with such tools and using them is not allowed. If, however, you wish to use such tools, ask for a more complex problem.

11   Provided files

- Validation calculation file
  - analyticalBlasius.py

12   Include in your report

- Running, parameterized code
  - The code should be able to accept velocity, viscosity, domain dimensions and mesh size as parameters
  - The code should automatically plot the results of your finite volume code in the same figure as the results from the validation code
  - It is required that you have written the code by yourself
  - include the code when sending the report. In a .py format

- Include velocity plots for both the analytical solution and your solution

- Mesh Independence study

- Governing equations

- Discretization methods

- Discretized governing equations
  - If you use the same methods for multiple terms, there is no need to repeat them all

13   Bag of tricks

- Use sparse matrices

- The flow will require some space to develop. Use a large enough domain.

- Remember to validate you code in as small steps as possible
  - During development, there will be bugs in the code. It is easier to spot them early.

References

[1] Fabian Peng Karrholm. Rhie-chow interpolation in openfoam, 2006. [Online; accessed 19-Feb-2018].

[2] R. Pitz and J. Daily. Experimental study of combustion in a turbulent free shear layer formed at a rearward facing step. 1981.

[3] H.K. Versteeg and W. Malalasekera. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method.* Pearson Education Limited, 2007.