

Simulating the fluid-structure interaction of a wind turbine blade in COMSOL

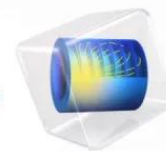
Authors Fabrizio Bruschetti
Josija Nydegger



Supervisor Prof. Dr. Eleni Chatzi
Assistants Dr. Imad Abdallah
Mr. Gregory Duthé

Date 27 June 2021

COMSOL
MULTIPHYSICS®



Acknowledgments

Our most important acknowledgements go to Prof. Dr. Eleni Chatzi, Dr. Imad Abdallah and Mr. Gregory Duthé. Without them all this would not have been possible.

They were great supervisors, always assisting us and being constantly available for all our questions and problems that arose in the process. We are very grateful that they also helped us find a solution when we didn't see one anymore.

We would also like to thank Mr. Roberto Pascolo, who enabled us to use the COMSOL software through a remote computer. We never had the chance to meet him, but for obvious reasons he was indispensable to us.

Finally, we must also sincerely thank Miss Valeria Jannelli and Mr. Noah Nydegger. Their linguistic help and corrections to the text of the report have been very important. Thank you very much.

Contents

1	Abstract	1
2	Introduction	2
3	Methodology	3
3.1	Water flow in a simple tube	3
3.2	2D and 3D tubes with complex geometry	5
3.3	NACA 0012 airfoil	8
3.4	Aventa airfoil	11
4	Results	15
4.1	Water flow in a simple tube	15
4.2	2D and 3D tubes with complex geometry	16
4.3	NACA 0012 airfoil	17
4.4	Aventa airfoil	20
5	Analysis	28
5.1	Water flow in a simple tube	28
5.2	2D and 3D tubes with complex geometry	29
5.3	NACA 0012 airfoil	30
5.4	Aventa airfoil	31
6	Discussion	35
7	Bibliography	37
8	Appendix	38

1 Abstract

In this Bachelor thesis, the airflow around a wind turbine blade is analyzed. For this purpose, the Multiphysics simulation software COMSOL was used. It provides very helpful tools that let us create a geometry, implement several functions that specify the fluid flow and finally extract the desired results. To familiarize ourselves with COMSOL, we conducted several tutorials. The final tutorial was called "*Flow Around an Inclined NACA 0012 Airfoil*" and it was provided by COMSOL (COMSOL, 2021 B). It was very helpful, since it was similar to the final model used to simulate the wind turbine blade. It contained step-by-step instructions on how to create the geometry, including the free space around the airfoil as well as how to create a mesh. For the first study, the model *Laplace Equation* was used to simulate the fluid-flow. Its results were then the starting point for the second study, which used a $k-\omega$ Turbulent Flow SST model to simulate the fluid-flow. The main results for this tutorial were pressure and velocity plots, as well as a graph with the pressure coefficient C_p for each angle of attack α .

Subsequently, the model was adapted to the new Aventa airfoil geometry. As a consequence, a new mesh had to be created, since the old one was specified for the old geometry and its boundaries. The results were very similar to the data Mr. Duthé provided. This comparison was necessary, since it confirmed that the model was actually providing the results we expected. Next, several input parameters were varied and the effects that these individual changes had on the results were observed. The inflow speed U_{inf} proved to not change the airflow or the streamlines; the pressure distributions were simply multiplied by a factor that stayed almost constant for a wide range of U_{inf} . As U_{inf} was increased, C_p was decreased.

The change of α had a much bigger impact on the airflow. As it was increased, the absolute pressure also increased, leading directly to a higher lift coefficient C_l . At the critical angle of attack of about 14° to 16° , the highest C_l is achieved. This critical α lies very close to the one for the NACA 0012 airfoil, which is between 17° and 18° (see analysis). For α s higher than the critical one, the airfoil starts to stall, indicated by a significant drop of the lift coefficient. Finding the right balance between having the most efficient energy production and the airfoil not stalling is a very important task.

As a next important step, critical α would therefore have to be verified by many more simulations with increased complexity such as the combination of different inflow parameters. Furthermore, with the introduction of fluid-structure interaction (FSI), it is possible to measure the effect the airflow has on the airfoil, and vice versa.

2 Introduction

Wind is the second largest renewable energy source after hydropower. In 2018, it produced more than 5% of global electricity (C2ES, 2021) and in 2016 it accounted for 16% of the electricity generated by renewables (IRENA, 2021). It is one of the fastest growing renewable energy technologies (IRENA, 2021). Its significance in the energy production is only increasing. Additionally, wind is not a static phenomenon but rather constantly changing in intensity and direction. It is therefore very important to design wind turbines that can adapt to these wind speeds and directions. Moreover, wind turbine blades continue to increase in size. This raises the chance of aeroelastic effects that are caused by fluid structure interaction (Wang, Quant, & Kolios, 2016). Even if the original size of the Aventa airfoil has only a chord length of 25 cm, simulating FSI still enables us to learn more about the interaction of a structure with its environment.

Especially because the wind turbine is not always under full load, its efficiency is crucial. The force that causes the turbine to rotate is the lift force on the blade, represented by the dimensionless lift coefficient Cl . As the results show, this lift coefficient grows as the angle of attack α is increased up to a certain value called critical angle of attack. After that, the airfoil begins to stall, resulting in a significant drop of Cl . To operate the wind turbine at its maximal efficiency, this critical α must be verified with numerous simulations. The inflow speed U_{inf} for example can be varied while varying α at the same time. Finding the α where the airfoil started to stall was indeed one of the main tasks.

The turbulent flow can be implemented in COMSOL with a range of turbulent flow models. For our simulations, the $k-\omega$ SST turbulence model was used, mainly because Mr. Duthé used it for his simulations and one COMSOL tutorial also used it. The SST (Shear Stress Transport) combines the superior behavior of the $k-\omega$ model in the near-wall region with the robustness of the $k-\epsilon$ model in the free stream, taking the best of both worlds (COMSOL, 2021 A), (SIMSCALE, 2021). It is a low Re model, meaning that it can be used to model the entire boundary layer, including the viscous sublayer and the buffer layer (COMSOL, 2017). The viscous sublayer is the region close to the wall where viscous effects dominate (see Figure 1).

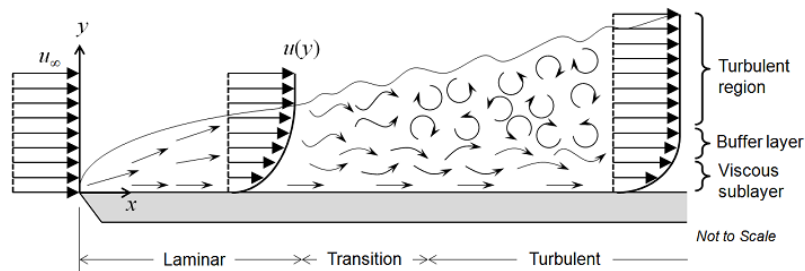


Figure 1: Fluid flow over a flat plate, the flow transitions from laminar to turbulent, while the vertical axis is divided into 3 different layers (COMSOL, 2017)

3 Methodology

3.1 Water flow in a simple tube

For the development of this model, a tutorial provided on the video platform YouTube was followed step by step (The Defeated Engineer, 2020 C)

First, it is necessary to select the type of model you want to build and the dimensional space of it: in this example we started a *model Wizard* in two dimensions.

Next, we were asked to pick the desired physics, selecting one from the many possible options. Laminar stationary fluid flow was selected, which represents a fluid whose values do not change in time.

At this point we arrive at the real *COMSOL* interface, where its components are observable and distinguishable: *Model Builder* (which summarizes and contains almost all the information/objects of the model), *Setting* (where it is possible to change and control the settings) and *Graphics* (where we can observe concretely what we are doing and the results that come out of it).

Let's now quickly summarize the different steps that led us to the creation of this small project:

- Creation of the geometry: fairly simple, it is composed only by four rectangles of different sizes in contact between them, clearly visible in Figure 2.
- Implementation of constraints/boundaries: after uniting the four rectangles (creating a single homogeneous body), it was necessary to select the *inlet wall* (top and bottom walls of vertical narrow rectangle) and the *outlet wall* (last right wall) of the liquid. Here, also the initial velocity of the material was set to 0.1 m/s.
- Addition of material: In this section we have chosen water. The program will therefore use all the characteristics and properties of it.
- Mesh creation: performed automatically by the program, no addition or modification was necessary. Its representation can be seen in Figure 3.

Once here, everything was ready for simulation computation, which led to some quick results that will be discussed in section 5.1.

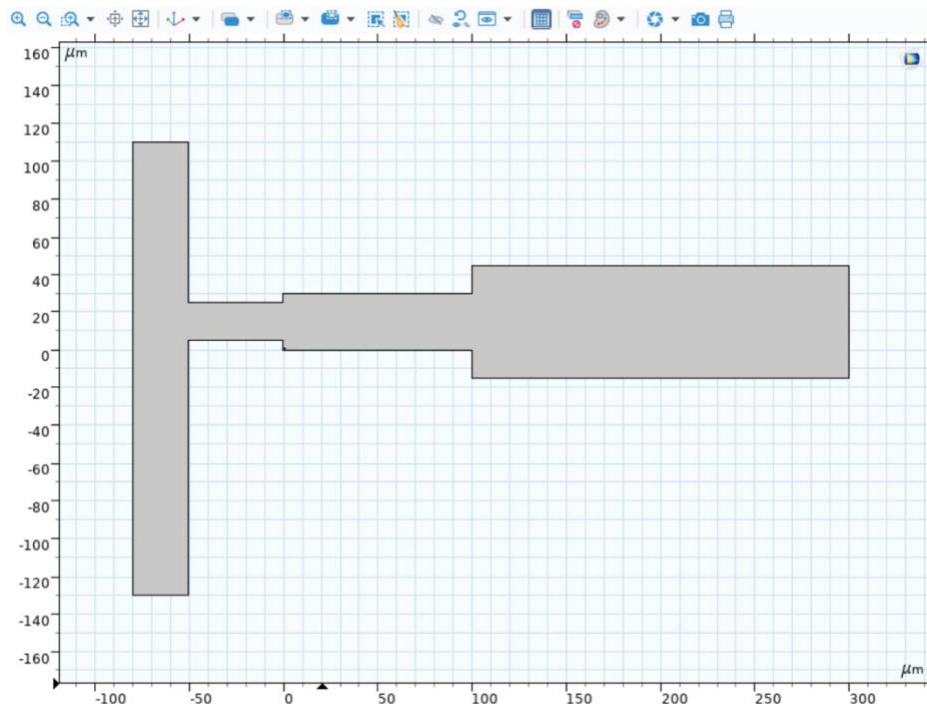


Figure 2: Geometry of the simple tube

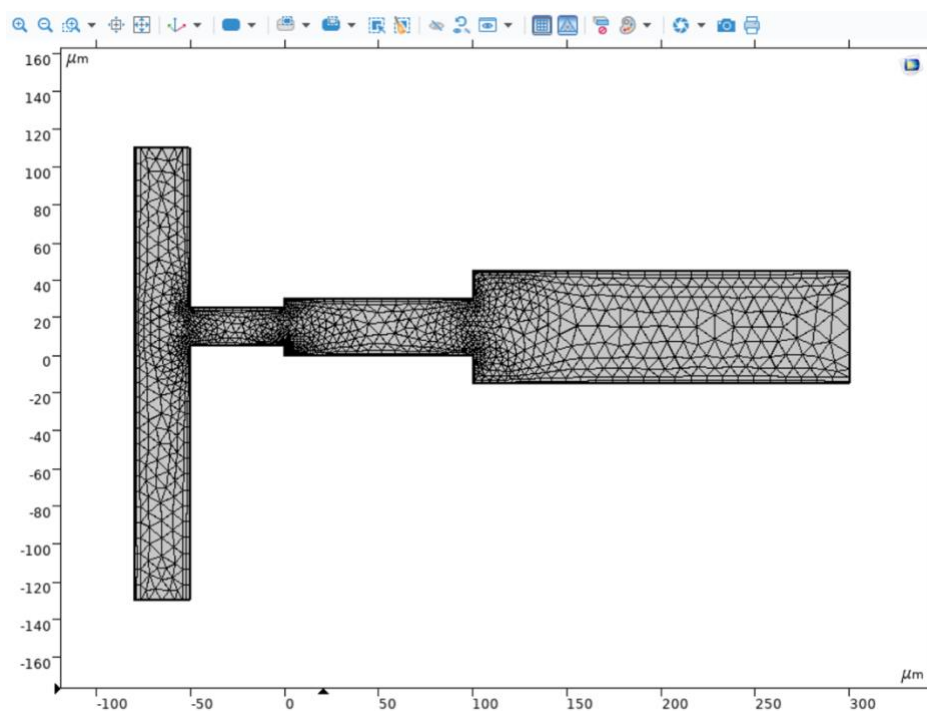


Figure 3: Mesh of the simple tube

For this first experiment, we wanted to explain almost all the steps in the creation of the model, step by step, as this was the aspect we wanted to focus on. For the next experiments, some stages of the construction will be omitted to avoid redundancy.

3.2 2D and 3D tubes with complex geometry

3.2.1 2D tube

Just as in the first experiment, we developed a model in two dimensions, which was quite simple but contained important innovations for us. The tutorial was again in the form of a YouTube video (The Defeated Engineer, 2020 A).

Thanks to the coordinates given in the tutorial, it was possible to create the first three necessary polygons consisting of eight segments attached to each other.

Subsequently, using the *mirror* method, we were able to duplicate one of the polygons already created, to quickly obtain the same polygon mirrored with respect to the *x*-axis obtaining the fourth 2D body.

At this point, the fundamental parts of the desired geometry were done and all that remained was to perfect it by adding small adjustments. Thanks to the *fillet* tool, we were able to round off all the desired corners of the design, creating more harmonious objects. Finally, thanks to the *union* command, we made the four components into a single object, eliminating the walls between them and consequently the constraints. All components of the model geometry are represented in Figure 4.

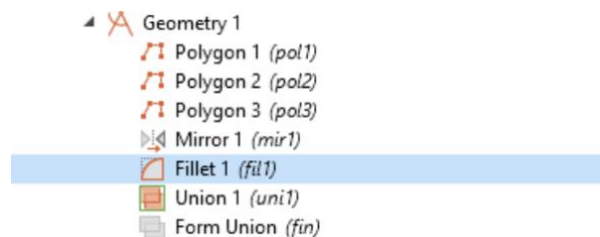


Figure 4: Geometrical components of the 2D tube

Now the model was geometrically complete: four different tubes, with rounded shapes, joined together at the center (see Figure 6). We had now reached our goal, which is why we stopped without further studying the model (we did not add any physics or materials, only the mesh illustrated in Figure 5).

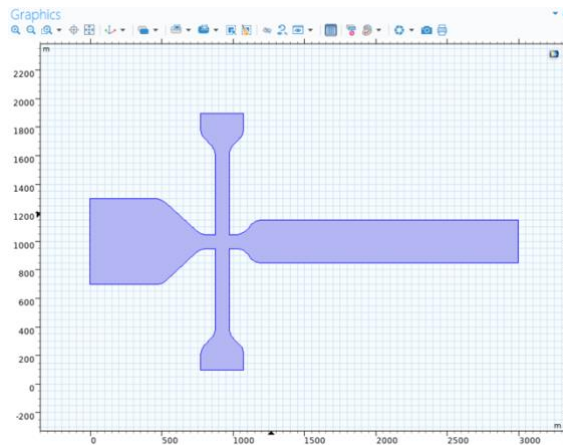


Figure 6: Final geometry of the model (2D complex tube)

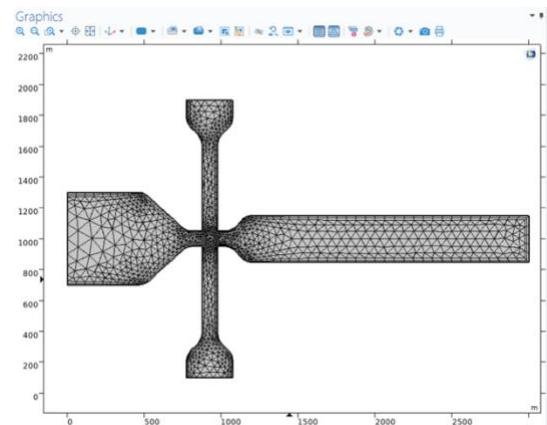


Figure 5: Mesh of the model (2D complex tube)

3.2.2 3D tube

This third tutorial was also in the form of a YouTube video (The Defeated Engineer, 2020 B) As indicated by the title, the major novelty of this last introductory experiment concerns dimensions: for the first time we worked in 3D. Actually, our final project runs entirely in two dimensions, which makes this tutorial not really necessary, but it was nevertheless of our interest to see new situations.

After designing a first cylinder (with dimensions and coordinates taken from the video), we placed two sections of a torus on one end, creating a double-ended tube. The torus sections went from 0° to 90° , as can be seen in Figure 8. Finally, as done several times already, we connected all the various bodies with the *union* function, creating a single object with no constraints/walls inside it. All components of the model geometry are represented in Figure 7.

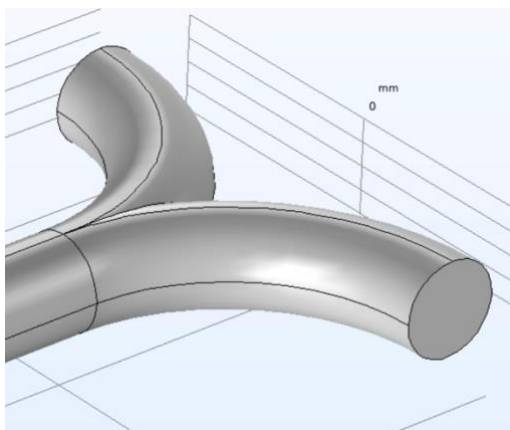


Figure 8: Torus from 0° to 90°



Figure 7: Geometrical components of the 3D tube

Again, the tutorial developed the model further by implementing physics, materials and studies, but this was not of particular importance to us. The final geometry and the most interesting parts of its mesh are shown in Figure 9, Figure 10 and Figure 11.

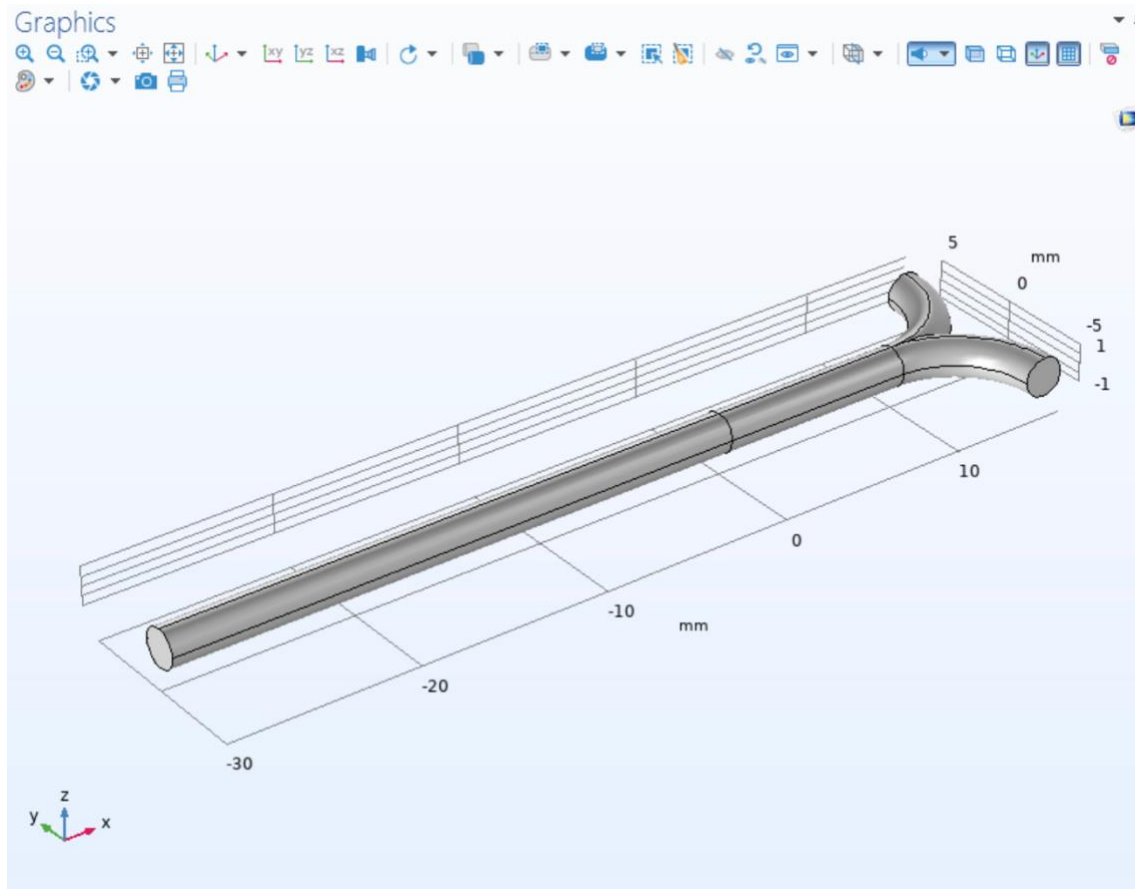


Figure 9: Final geometry of the model (3D complex tube)

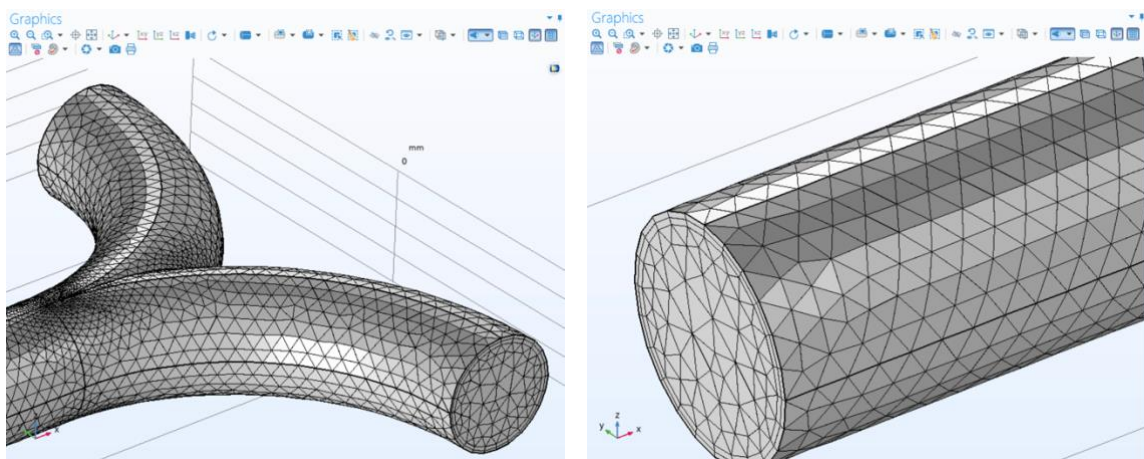


Figure 11: Mesh of the model (Final Separation Section) Figure 10: Mesh of the model (Initial 3D tube section)

3.3 NACA 0012 airfoil

COMSOL provides numerous tutorials with small examples for each model. These tutorials consist of a PDF document with step-by-step instructions to build the model from scratch. Additionally, to compare the obtained results, a completed COMSOL model is provided.

The introduction given directly from the PDF provided by COMSOL regarding this experiment says:

“This example simulates the flow around an inclined NACA 0012 airfoil using the SST turbulence model and compares the results with the experimental lift data (...). The SST model combines the near-wall capabilities of the $k - \omega$ model with the superior free-stream behaviour of the $k - \varepsilon$ model to enable accurate simulations of a wide variety of internal and external flow problems.” (COMSOL, 2021 B)

This tutorial represents a perfect point of departure for the creation of our project, containing so many characteristics and features similar to the model that we want.

The first important thing to do was to define the values on which to build the whole model. These were given to us directly from the pdf of the tutorial and are listed in the Figure 12.

Name	Expression	Value	Description
U_inf	50[m*s ⁻¹]	50 m/s	Free-stream velocity
rho_inf	1.2043[kg*m ⁻³]	1.2043 kg/m ³	Free-stream density
mu_inf	1.81397e-5[kg*m ⁻¹ *s ⁻¹]	1.814E-5 kg/(m*s)	Free-stream dynamic viscosity
L	180[m]	180 m	Domain reference length
c	1.8[m]	1.8 m	Chord length
k_inf	0.1*mu_inf*U_inf/(rho_inf*L)	4.184E-7 m ² /s ²	Free-stream turbulent kinetic energy
om_inf	10*U_inf/L	2.7778 1/s	Free-stream specific dissipation rate
alpha	0	0	Angle of attack

Figure 12: Parameters of the NACA 0012 experiment

Now it was time to define the necessary geometries. This part can be divided into two parts: on one hand we had to build the flow domain and the applied far-field boundary conditions, on the other hand the airfoil section and its constraints. In both cases, it was a simple and fast operation, since the flow domain had very basic geometries (circles and rectangles joined together) and the airfoil section is quickly created thanks to the given functions:

$$x = c \cdot s$$

$$y = c \cdot 0.594689181 \cdot (0.298222773 \cdot \sqrt{s} - 0.127125232 \cdot s - 0.357907906 \cdot s^2 + 0.291984971 \cdot s^3 - 0.105174696 \cdot s^4)$$

The final geometries are shown in Figure 13 and Figure 14, where the second is a close-up of the first.

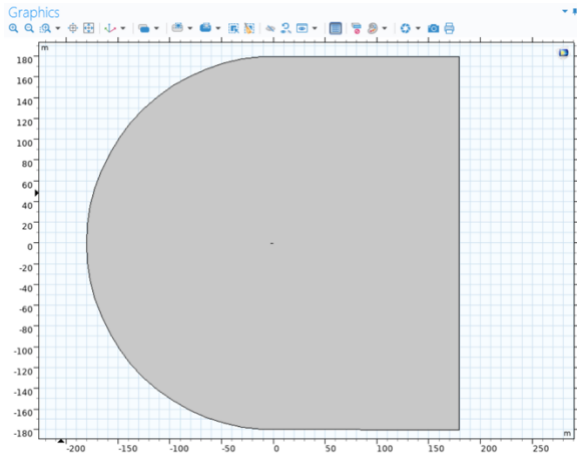


Figure 14: Flow domain NACA 0012 experiment

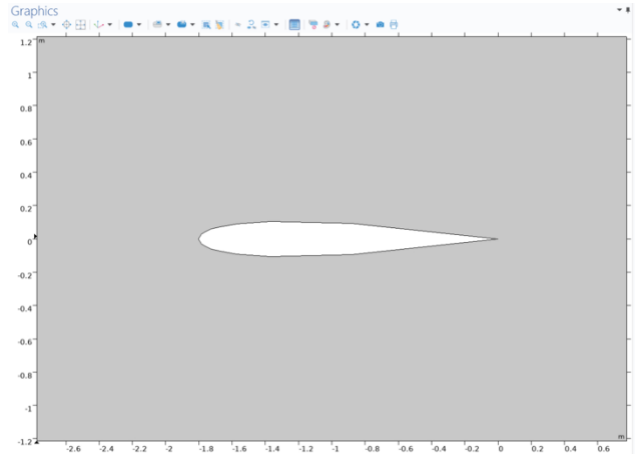


Figure 13: Airfoil section NACA 0012 experiment

After that, it was time to define the material. In this case too it was very straightforward because, after selecting the material *air*, COMSOL recognizes all the characteristics and values that describe it.

Next, boundaries were defined: As can be seen from Figure 15, Figure 16 and Figure 17, we have defined as *inlet* all the curved walls on the left, as *open boundary* the entire vertical wall on the right and as *wall* the surface of the airfoil.

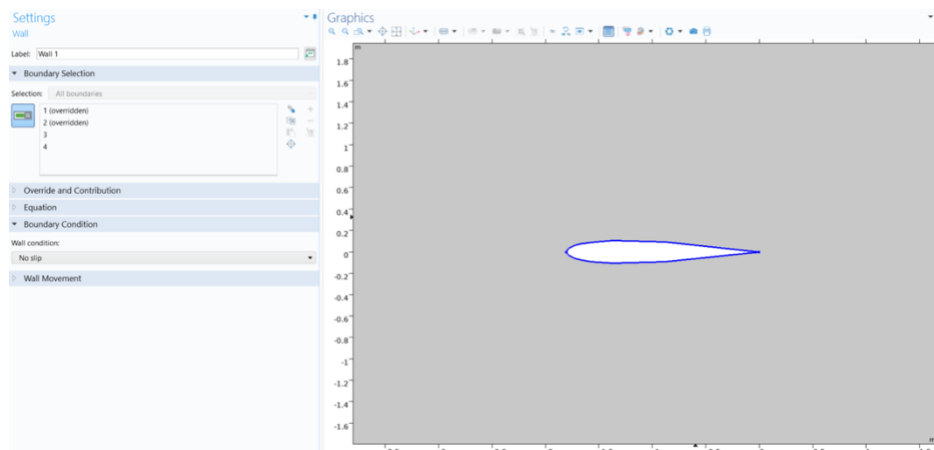


Figure 15: Wall representation and parameters of the NACA 0012 experiment

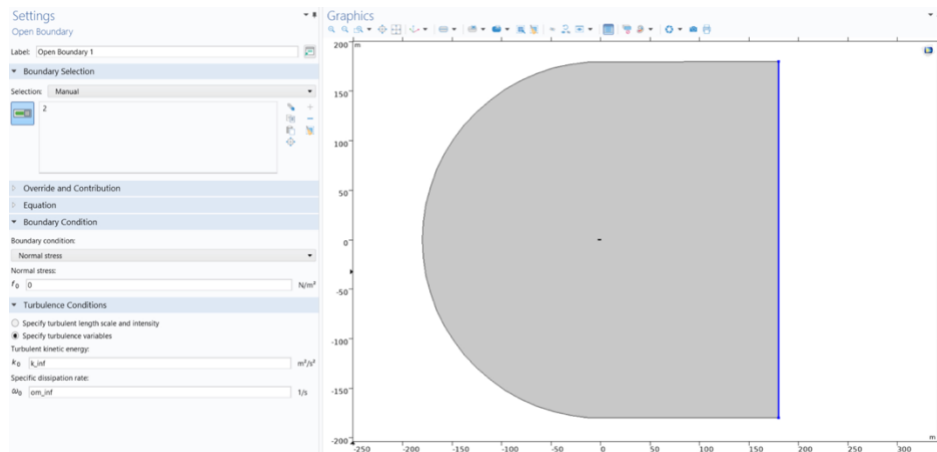


Figure 16: Open boundary representation and parameters of the NACA 0012 experiment

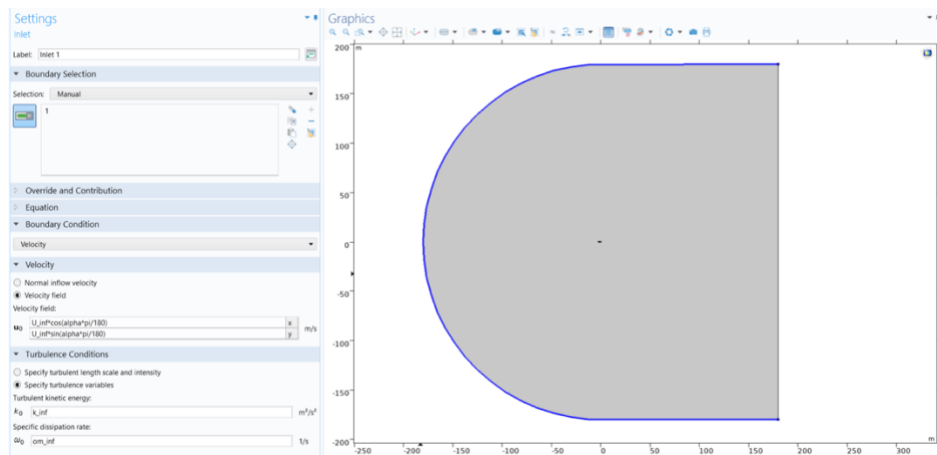


Figure 17: Inlet representation and parameters of the NACA 0012 experiment

We now come to the last necessary step before we can get results from the created model: the creation of the mesh. Unlike what we did until now (see section 3.1 and 3.2), we did not use the auto-mesh that COMSOL proposes, but we created one manually. The tutorial presented in fact a particular modeling of the mesh, rather complex and extensive but certainly useful for the success of the experiment. Because there are a lot of steps involved in creating the mesh, instead of listing them all, we directly show the final obtained mesh. It is visible in Figure 18 and Figure 19.

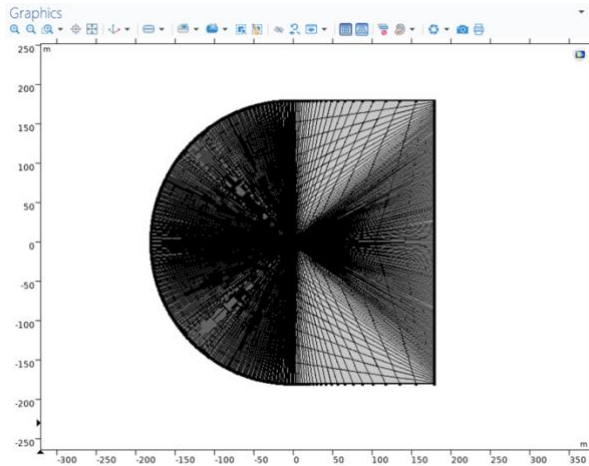


Figure 18: Mesh for the NACA 0012 experiment

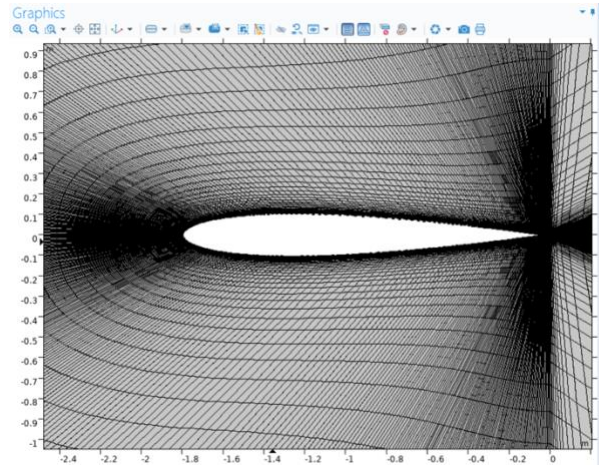


Figure 19: Close-up of the Mesh for the NACA 0012 experiment

At this point, our model has everything in place: once we set up the studies, we have indeed received several results, which will be presented in Section 4.3.

We found it important to describe the creation of this tutorial in enough detail because many aspects and different geometries are the same as those of our final project that follows now.

3.4 Aventa airfoil

3.4.1 Introduction of Aventa airfoil geometry

In the following, the COMSOL model was adapted by introducing the geometry of the Aventa airfoil. Mr. Gregory Duthé provided this geometry as coordinates in a *dat* data file. The x- and y-coordinates were then added as a polygon with solid shape. Because the airfoil was no longer symmetrical, the complete flow domain had to be rebuilt as well. After inserting the circular section and the square, the shapes were mirrored and combined as a union. Finally, a difference was created to separate the shape of the airfoil from the rest of the domain.

Mr. Duthé also provided the updated input parameters and some background information of the actual windmill:

In reality, the airfoil has a chord length of 25 cm and is at the tip of the test wind turbine. The turbine has a 12.9 m diameter rotor and at a wind speed of 7 m/s it achieves a rated power of 6.5 kW and spins at 66 RPM. With the relative velocity at the tip of the blade of 44.5 m/s and air temperature of 20 °C, the Reynolds number can be calculated to

$$Re = 7.4 \cdot 10^5$$

For the model, a normalized chord length of 1 m was chosen, which resulted in an inflow velocity $U_{inf} = 11.18 \text{ m/s}$.

3.4.2 Adding the new Mesh

Due to the changes in geometry, the mesh had to be recreated. This was done using a physics-controlled mesh provided by COMSOL. The element size was set to *normal*. After selecting *Build All*, the mesh was built without any additional settings needed. As seen in Figure 20, the mesh met the requirements, namely that it becomes finer as it approaches the surface of the airfoil.

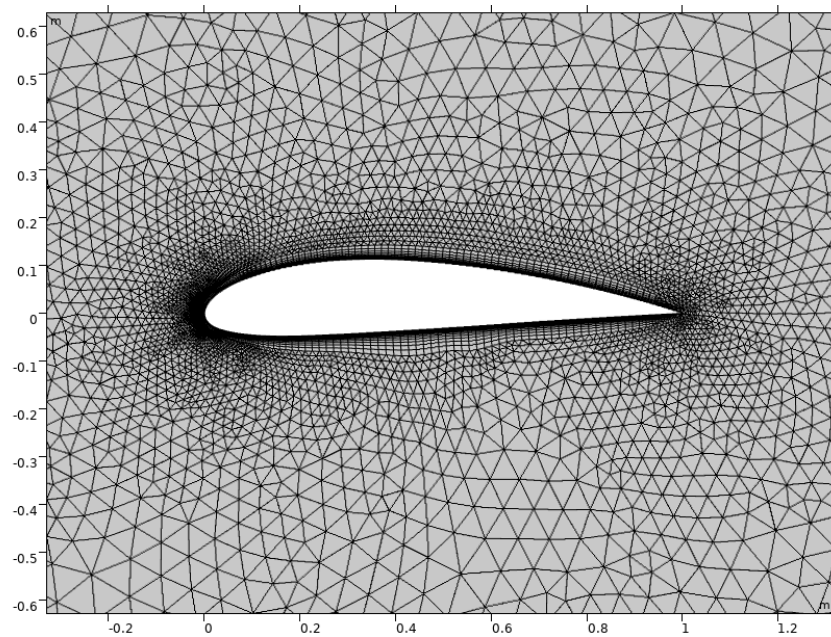


Figure 20: Zoomed in mesh, scale in meter

3.4.3 Adjusting the model for various outputs

The tutorial from COMSOL already included a method to calculate the pressure coefficient C_p : in the section *Line Graph I* the y-axis data is calculated with the following expression:

$$-p / \left(\frac{1}{2 \cdot \rho_{inf} \cdot U_{inf}^2} \right)$$

This results in a negative C_p . To compare the C_p distributions with the Xfoil data, this term was used without negative sign. Furthermore, for the x-axis data the tutorial used the following expression:

$$\frac{(x + c)}{c}$$

Again, to compare the data, the term was changed to x/c so that it matches the one Mr. Duthé used.

Furthermore, we found out that COMSOL uses the unit radians for angle α (see chapter 3.4.5). All the expressions using $\frac{(\alpha \cdot \pi)}{180}$ were therefore changed to use just (α) .

The *Line Graph 1* was then exported as *Plot 1* under the section *Export*. The C_p data is represented in a two-column table and saved as a *txt* text file.

For export, the pressure is presented in a surface plot similarly to the velocity plot instead of a contour plot as the tutorial intended. The tool *Image Snapshot* was used to export the velocity and pressure plot as detailed as possible.

3.4.4 Variation of U_{inf}

As a first modification of the model, the inflow speed U_{inf} was varied. Starting from 11.18 m/s, U_{inf} was increased in steps of 1 m/s up to 16.18 m/s. Because the change of speed doesn't affect the direction of the airflow, the solution was updated respectively instead of recomputing the whole solution after every increment. This resulted in significantly lower simulation time and thus a more straightforward process.

3.4.5 Variation of α

First, the parameter U_{inf} was set back to its initial value of 11.18 m/s. Subsequently, the angle of attack α was increased from 0° up to 20° in steps of 2° . Because this variation of α directly changes the flow of air, the two Studies containing the Laplace Equation and Turbulent Flow had to be completely recalculated for every increment. This resulted in much higher computation time of about 3 minutes.

Unlike mentioned in the tutorial, COMSOL already converts degrees to radians and therefore, multiplying the value by $\pi/180$ results in very low alphas and thus not correct solutions.

After updating all the terms that include α to just use (α) instead of $(\alpha \cdot \pi/180)$ the velocity plots clearly showed that the model now uses the correct angle of attack.

For $\alpha = 6^\circ, 8^\circ$ and 10° , the computation took even longer and eventually ended in an error. The error mentioned a division by zero which resulted in the model failing to evaluate several expressions (see analysis).

After a discussion with Mr. Duthé, several measures were taken to solve the problem by making the solution converge: the element size of the mesh was set to *coarse* and the *number of iterations* under *solver configurations > Stationary Solver 2 > Segregated 1* was increased to 500 and later 600. These measures at first didn't eliminate the error.

Finally, setting the *number of iterations* to 200 resulted in a successful calculation of the pressure values and plots required.

As seen in the Figure 30, the C_p plots for $\alpha = 8^\circ$ and 10° differ greatly from the other angles. For this reason, they were not included in the general C_p diagram.

Lastly, negative α s from -2° to -8° were used to compare the pressure and velocity of negative and positive angles.

3.4.6 Variation of k_{inf}

As the last parameter, the Free-Stream turbulent kinetic energy k_{inf} was varied. First, it was multiplied by a very small and very large number to see if the variation even had an effect on the pressure distribution. The initial parameter k_{inf} was multiplied by the factors 0.001 and 10'000 to cover a wide range. The pressure changed only minimally, and the change wasn't visible in the plots (see Excel file). Due to this, no further simulations were carried out for k_{inf} .

3.4.7 Lift coefficient vs. angle of attack

As explained in full detail in the NACA 0012 tutorial, the lift coefficient Cl was calculated using following integration:

$$C_L(\alpha) = \oint_c (c_p(s)/c) (n_y(s)\cos(\alpha) - n_x(s)\sin(\alpha)) ds$$

where the pressure coefficient is

$$c_p(s) = \frac{p(s) - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

This Cl is already adjusted for the angle of attack α with the trigonometric functions (Lyu, 2015). The function *Line Integration* under *Results > Derived Values* was used to integrate the expression.

The NACA 0012 tutorial included a method that used the function *parametric sweep* to directly calculate Cl for different α s and plot them in a graph. For the new model with updated geometry, this *parametric sweep* was not used and therefore, the lift coefficient was manually extracted from the model for each α and added to the Excel file to create the graph shown in Figure 42.

4 Results

4.1 Water flow in a simple tube

As already mentioned, the main objective of this tutorial was to become familiar with the program and begin to understand the process necessary to create a model. We could therefore say that the most important result is just having become acquainted with COMSOL and several of its facets useful to our bachelor's thesis.

However, by carefully following the tutorial, we were able to obtain some interesting data and plots. More precisely, we analyzed the graphs representing the velocity distribution (Figure 21) and the pressure distribution (Figure 22) inside the tube during the passage of water.

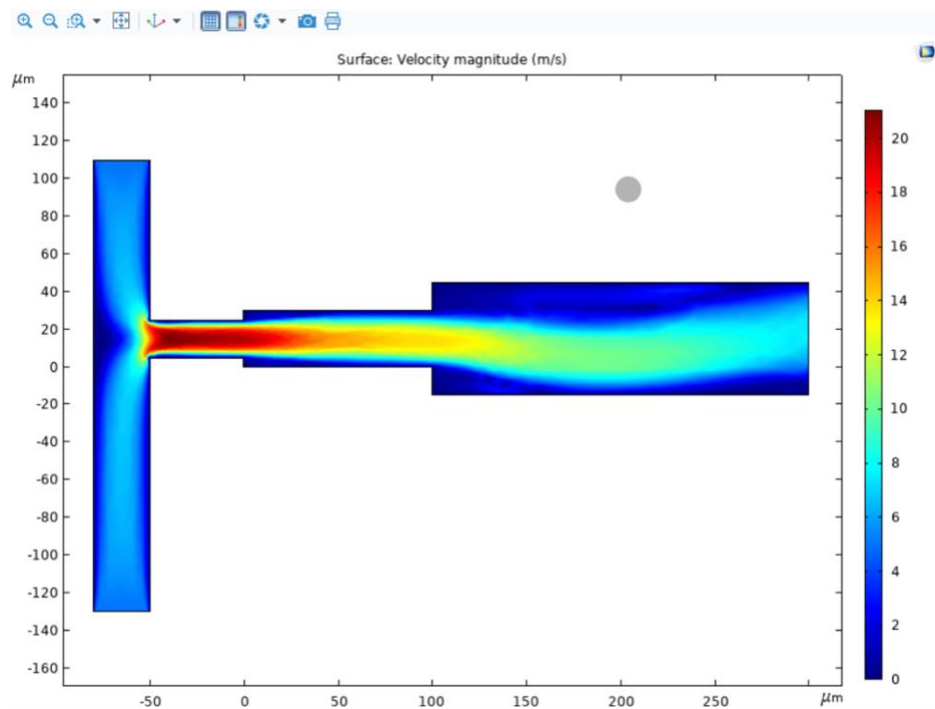


Figure 21: Velocity distribution of water flow in a simple tube

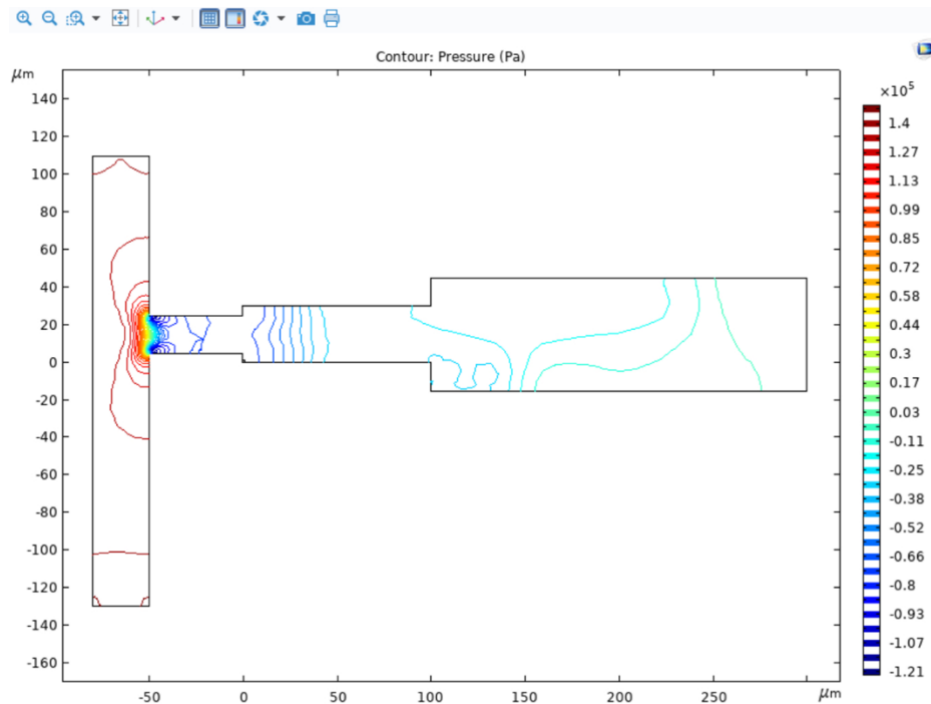


Figure 22: Pressure distribution in a simple tube

With regard to the velocity of the fluid, it can be seen that we pass from areas with almost no velocity (where the coloring is dark blue), to others with a maximum velocity of approximately 85 to 90 m/s (fiery red parts).

If instead we look at the pressure plot, we can distinguish maximum values of about $1.5 \cdot 10^5 \text{ Pa}$ (dark red lines) and minimum values of about $-1.2 \cdot 10^5 \text{ Pa}$ (dark blue lines).

4.2 2D and 3D tubes with complex geometry

For both experiments we do not have much to say in terms of results. As we have repeated many times, we had no interest in adding studies/physics to the model, as these were simple "training exercises". The most important result was to become even more familiar with COMSOL, learning to shape geometrically complicated bodies and to use tools previously unknown to us.

What came out of these tutorials has already been presented in the sections 3.2.1 and 3.2.2, where we presented the final geometries of the models (see Figure 6 and Figure 9).

4.3 NACA 0012 airfoil

The most important and interesting results to be discussed are the following four: the representation of the flow velocity distribution (with $\alpha = 14^\circ$), the representation of the pressure distribution (with $\alpha = 14^\circ$), the graph α vs. Cl and the graph $\left(\frac{x-x_{LE}}{c}\right) / (-cp)$.

All the results just mentioned can be seen from Figure 23 through Figure 26.

First, we want to observe the velocity and the pressure plot.

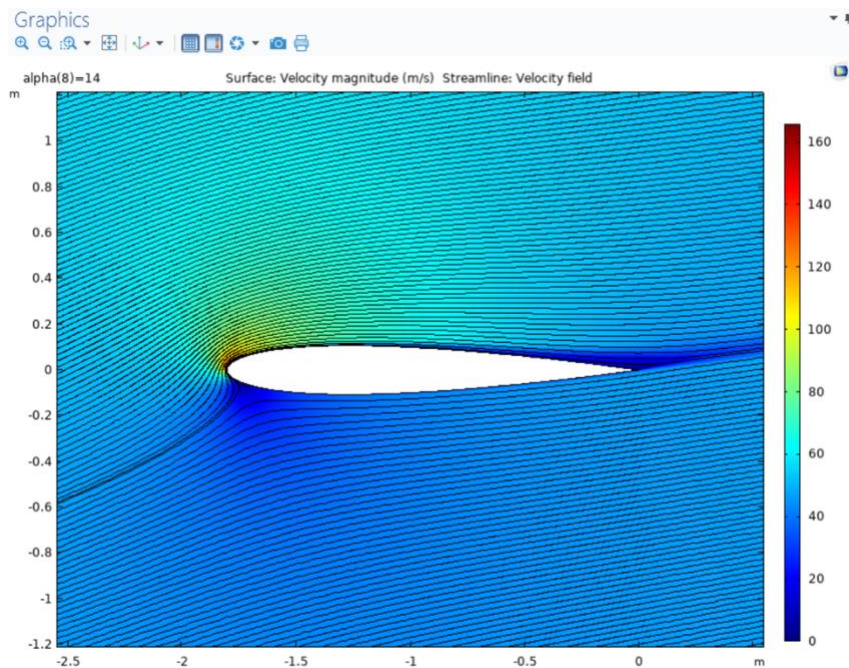


Figure 23: Flow velocity distribution with $\alpha = 14^\circ$

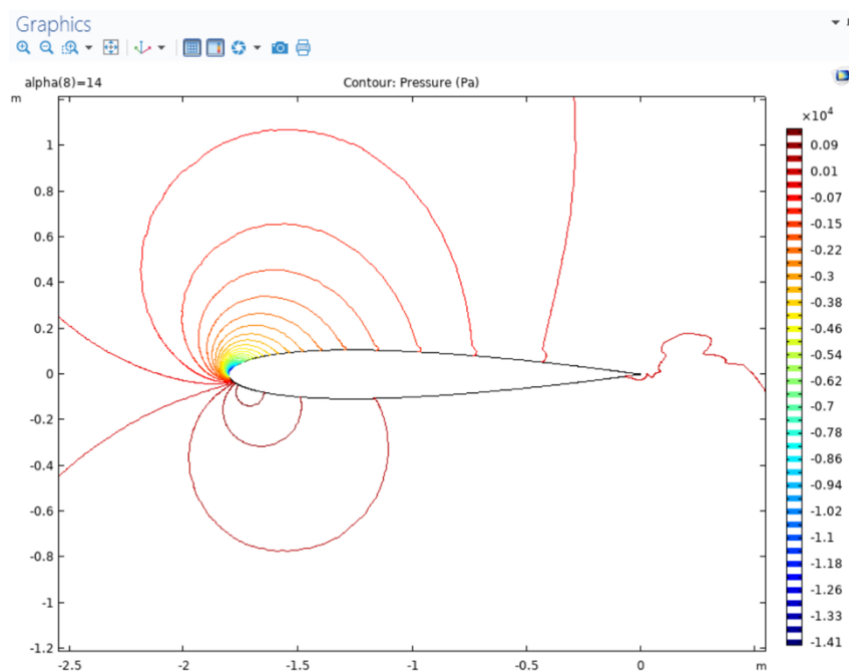


Figure 24: Pressure distribution with $\alpha = 14^\circ$

From the first, where we set an initial flow velocity $U_{inf} = 50 \text{ m/s}$, four aspects are quickly noted:

- The impact zone with the airfoil section perpendicular to the flow has a velocity close to zero.
- The final area of contact with the airfoil (top of the final tip) also exhibits a near-zero flow velocity.
- The incoming flow, initially placed by us at $\alpha = 14^\circ$, tends to straighten out as it passes the airfoil section. In fact, the outgoing flow is certainly less inclined than the incoming flow, and we estimate it to have a maximum value of 10° .
- We observe the maximum velocity at the top of the left curvature, i.e. at the place where the slope of the curvature approaches the inclination angle of the airflow. In this area we arrive at values around 150 m/s .

If we now look at the pressure plot, we notice other notable characteristics:

- In the perpendicular impact section (where we saw earlier we have one of the minimum velocity values) and in general in almost the entire lower surface of the airfoil we find the maximum pressure values. Here we observe peaks of about $(0.05 \cdot 10^4) \text{ Pa}$.
- On the other hand, in the area where we have the maximum speed of the flow (as mentioned before, the upper part of the curvature on the left) we also have the minimum pressure values. Here, we reach very negative numbers, which are around $(-1.4 \cdot 10^4) \text{ Pa}$.
- In general, the dominant value is that of atmospheric reference pressure of 0 Pa .

Now let's look at the other two graphs mentioned above.

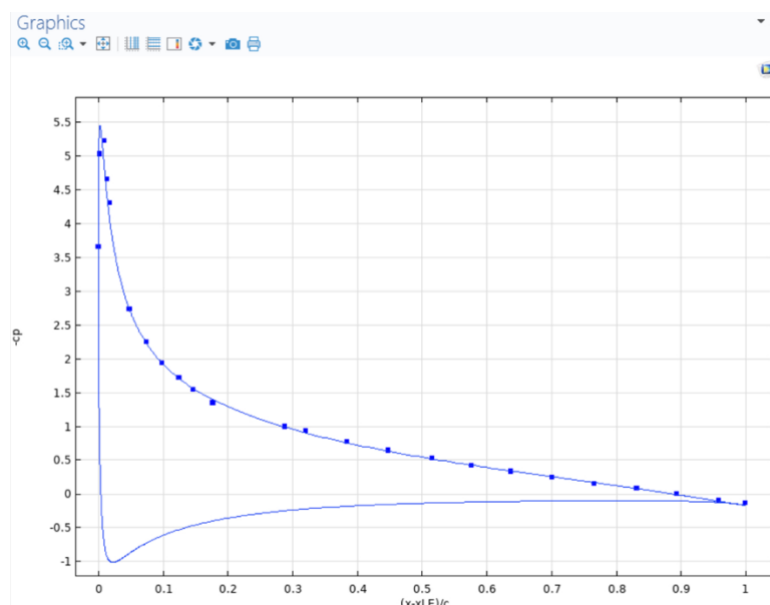


Figure 25: C_p distribution around the airfoil

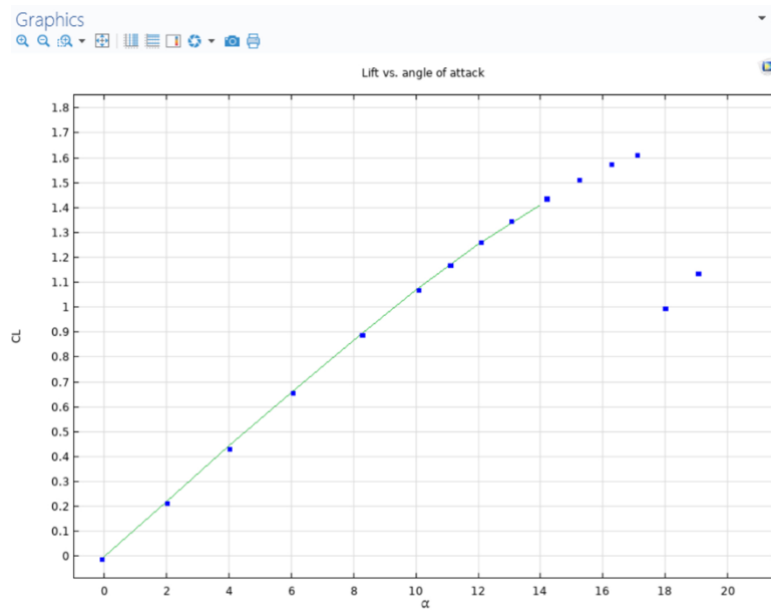


Figure 26: Lift coefficient as a function of the angle of attack

The graph in Figure 25 represents the pressure distribution (in the form of a negative scalar) around the entire perimeter of the studied structure, diversifying the top and bottom of the airfoil with two lines. It can be quickly seen that the most negative value is about -5.5, while the most positive value is close to 1. We will analyze later the meaning of such scalars, which give interesting information about our model.

Finally, we could also reproduce the graph of the Lift-coefficient as a function of the angle of attack, visible in Figure 26. Here, we observe an almost linear trend: up to values of about $\alpha = 12^\circ$, the plot shows a straight line with positive slope starting from the origin. The graph tends to curve at the end, decreasing steadily (and slightly) its slope. This is due to the airstream gradually detaching from the surface of the airfoil. The highest C_l of about 1.6 is observed for $\alpha = 17^\circ$. After this, the significant drop of lift coefficient indicates that the airfoil stalls. More on this phenomenon in the next chapter.

4.4 Aventa airfoil

As mentioned earlier, the model exports the C_p values as a two-column table in a *txt* file. Thus, for every horizontal value x/c , there are two C_p values for the upper and lower surface of the airfoil. In contrast, the data Mr. Duthé provided in the Excel file is already separated into two parts using a third column with the vertical y/c values.

It was therefore not possible to connect the points with a line because the points alternate between upper and lower surface. Hence, the single points were represented by small 2pt squares that give the plot the appearance of a continuous line.

A line at $C_p = 0$ was added to better identify the points where C_p turns from negative to positive and vice versa.

Figure 27 shows the C_p data from our COMSOL model compared to two other distributions: Firstly, computational Fluid Dynamics (CFD) without fluid-structure interaction and secondly a C_p distribution given by Xfoil, a potential flow solver designed specifically for airfoils. For the Xfoil model, the same inflow parameters such as inflow speed and angle of attack were used as with the COMSOL model. A k-omega SST simulation was run.

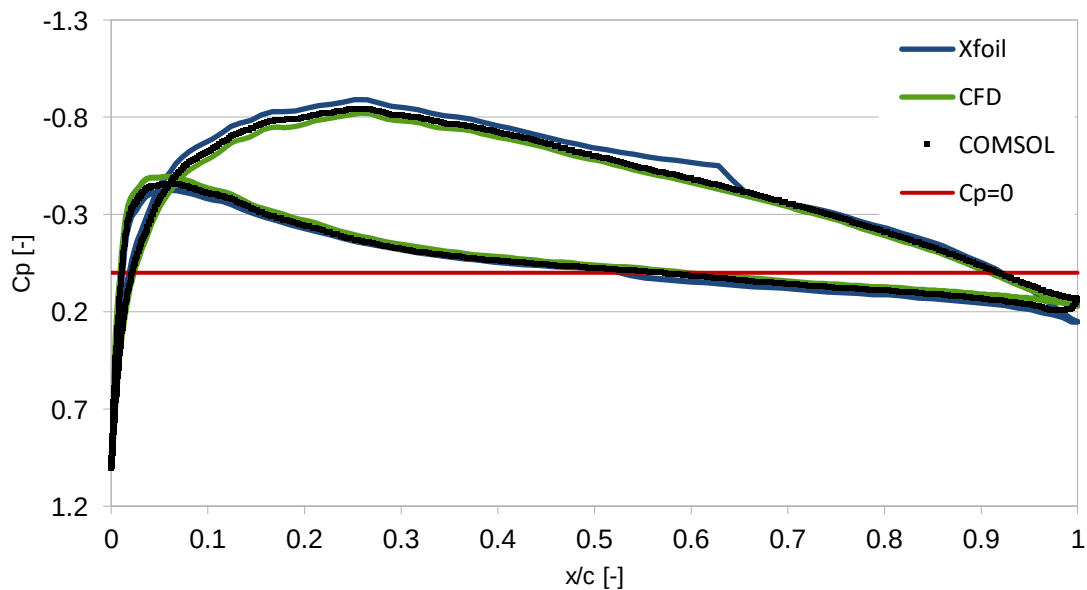


Figure 27: Pressure coefficient C_p from COMSOL compared to other simulations

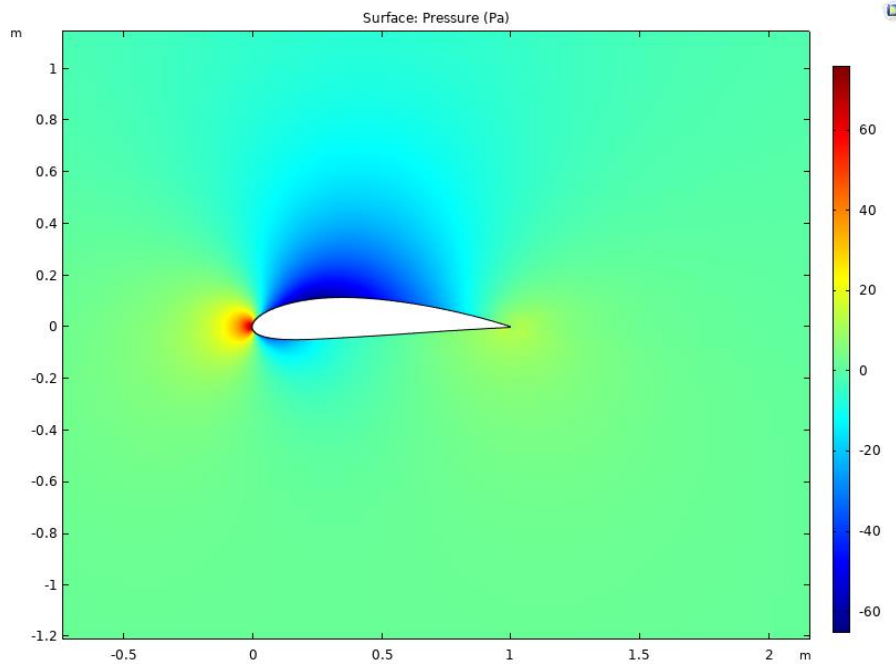


Figure 28: Pressure magnitude for $\alpha = 0^\circ$

Figure 28 shows the pressure distribution for $\alpha = 0^\circ$. The pressure is negative for the upper surface and part of the lower surface, while it is highly positive at the tip of the airfoil.

4.4.1 Variation of U_{inf}

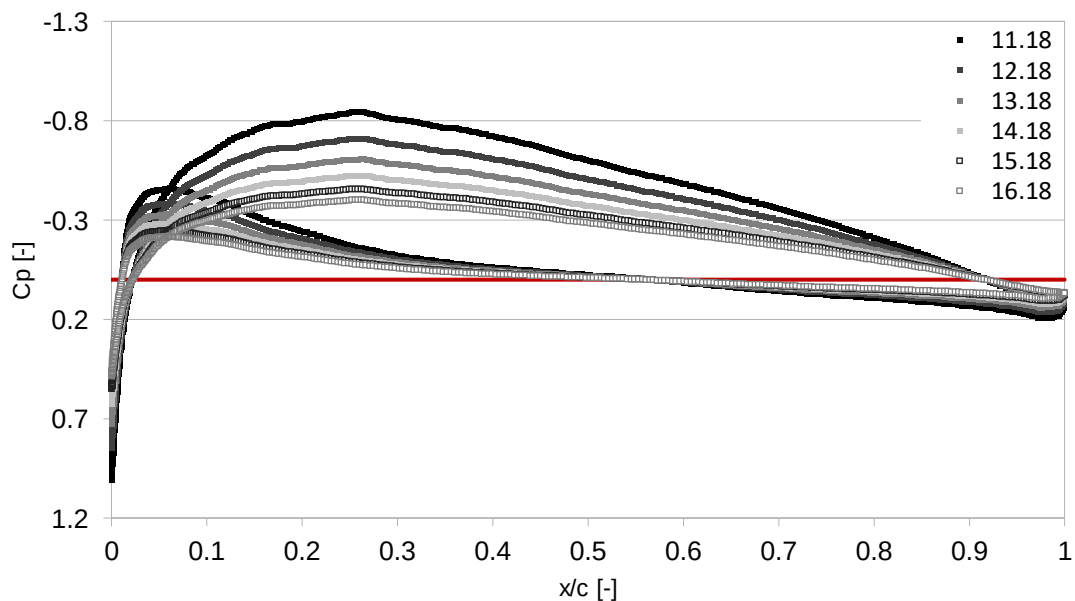


Figure 29: Pressure coefficient C_p for increasing inflow speed U_{inf} [m/s]

Figure 29 shows the pressure distribution for increasing inflow speed U_{inf} . With increasing U_{inf} , the absolute pressure decreases.

4.4.2 Variation of α

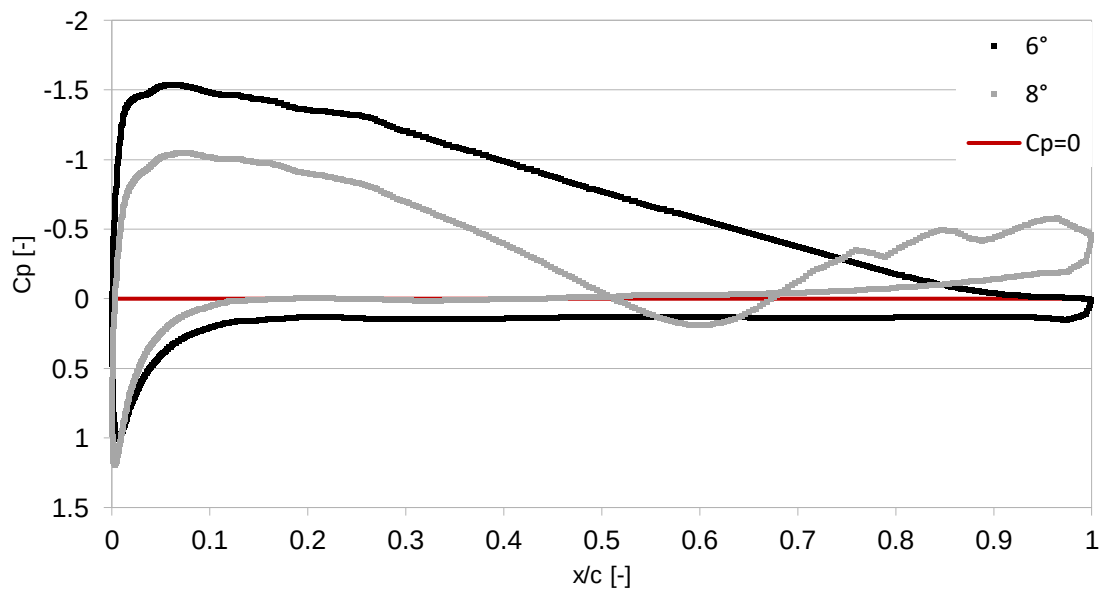


Figure 30: Pressure coefficient C_p for $\alpha = 6^\circ$ and $\alpha = 8^\circ$

As seen in Figure 30, the pressure for $\alpha = 8^\circ$ differs greatly from the one for $\alpha = 6^\circ$, especially on the second half of the upper surface. As further explained in the discussion, we assume this is due to a strongly turbulent flow in the rear part of the airfoil. Figure 31 confirms this assumption: There are great fluctuations in speed in a comparably small area. The velocity plot for $\alpha = 10^\circ$ shows the same phenomenon. For both angles, this anomaly might also be the result of the simulation not terminating correctly due to the low *number of iterations* of 200. The corresponding pressure distributions might therefore not be the true results for the respective angles of attack. They were therefore omitted in the following figures.

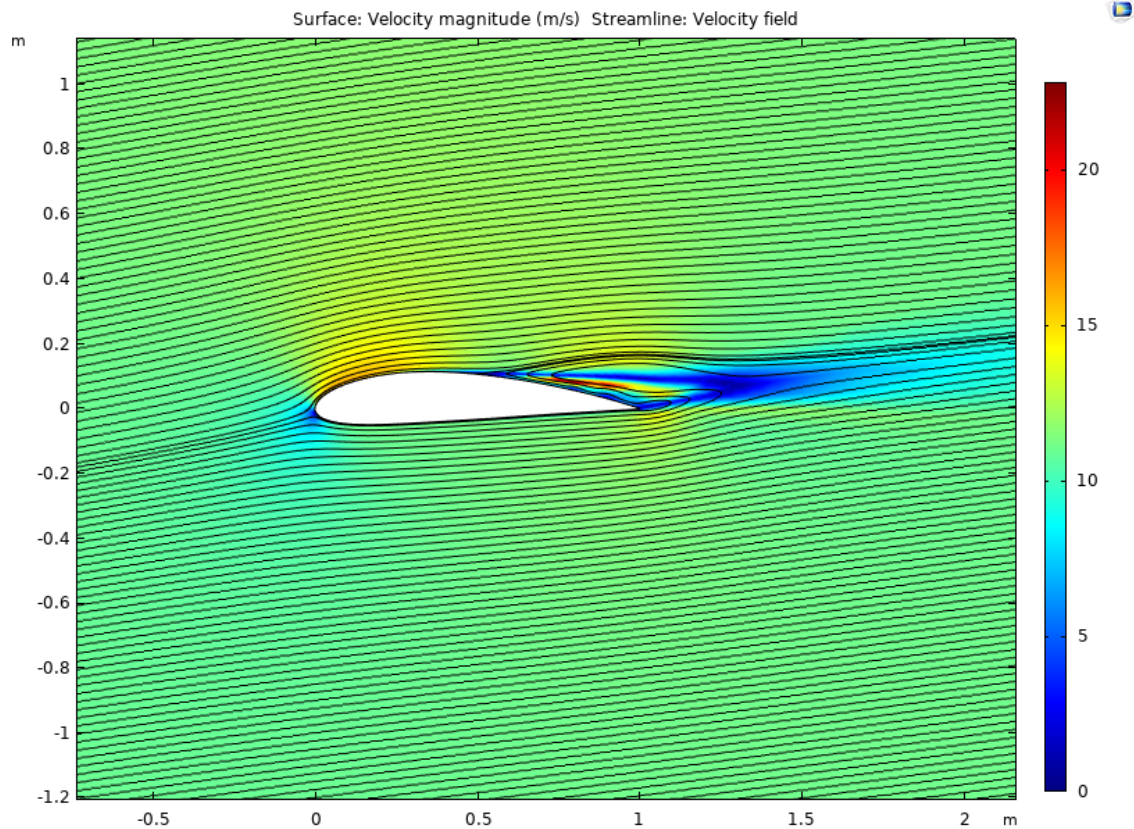
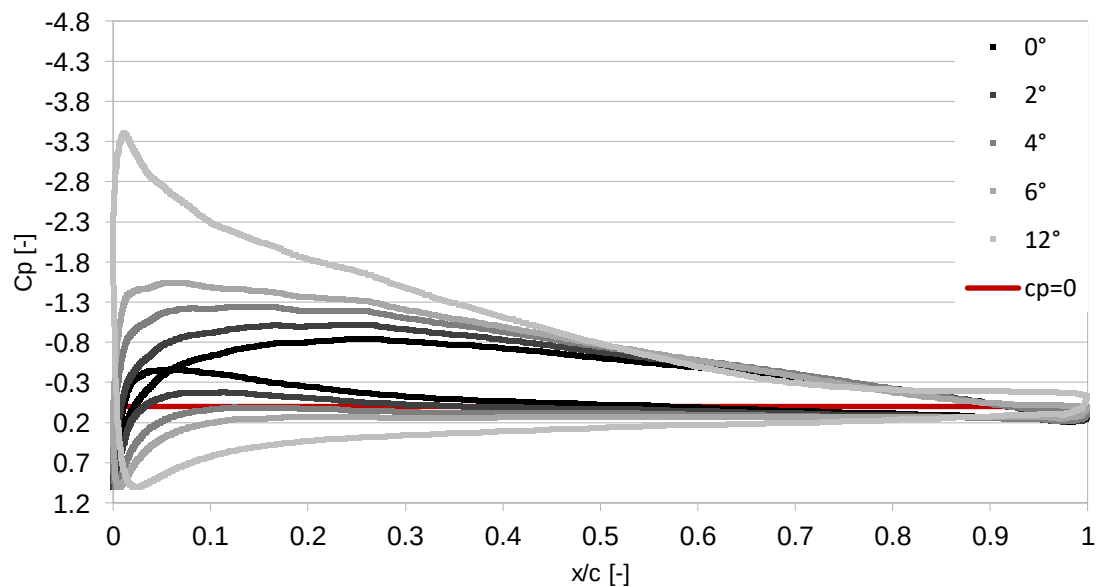
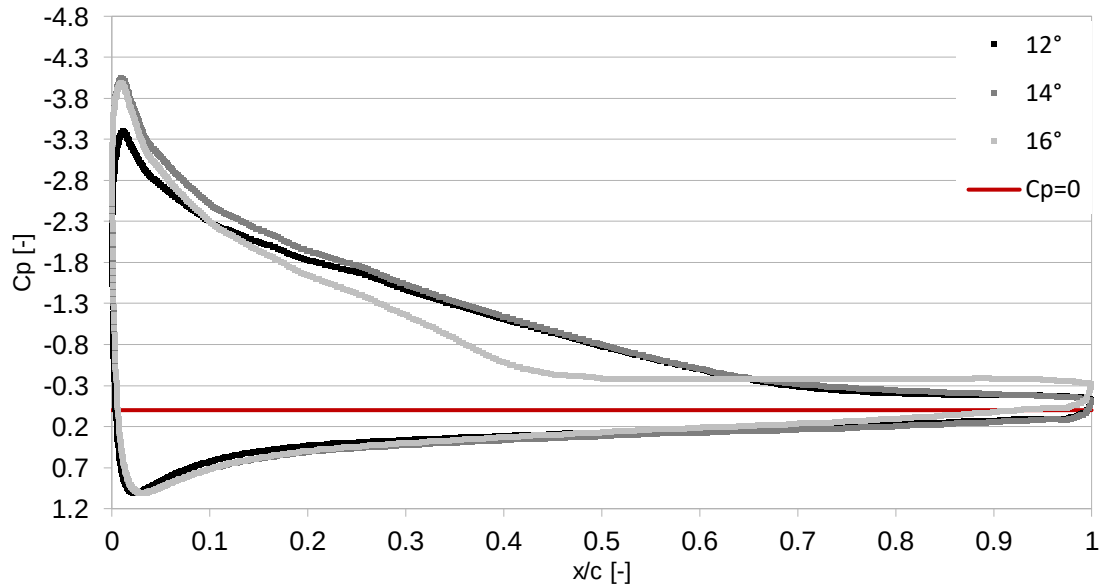
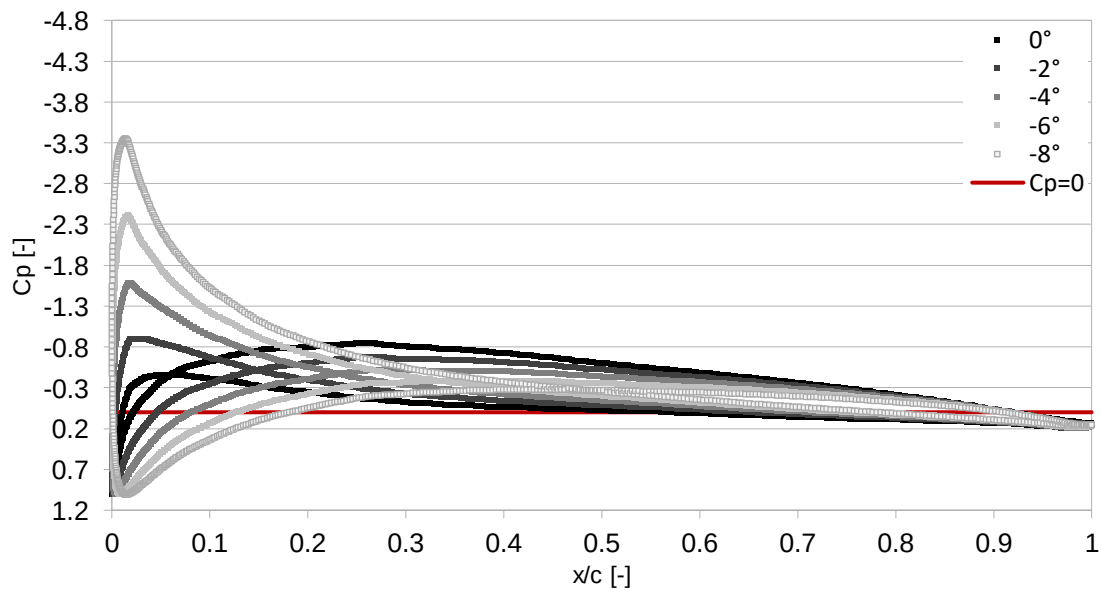
Figure 31: Velocity magnitude and streamlines for $\alpha = 8^\circ$ Figure 32: Pressure coefficient C_p for $\alpha = 0^\circ$ to $\alpha = 12^\circ$, omitting 8° and 10°

Figure 32 shows the pressure distribution for for $\alpha = 0^\circ$ to $\alpha = 12^\circ$. As α increases, the absolute pressure increases too.

Figure 33 shows the pressure distribution for higher α s. The curves for $\alpha = 12^\circ$ and 14° are very similar, while the one for $\alpha = 16^\circ$ differs more.

Figure 33: Pressure coefficient C_p for higher alphas 12° to 16°

As mentioned in section 3.4.5, α was increased up to 20° . After a discussion with the assisting supervisors, α s higher than 16° were not included in the plots, because such high angles of attack are on one hand not realistic for a wind turbine airfoil and on the other hand the used SST turbulence model is not suited for these angles. The produced pressure distribution would not allow any reasonable discussion. Furthermore, as discussed later, the airfoil starts to stall at around $\alpha = 16^\circ$.

Figure 34: Pressure coefficient C_p for negative alphas 0° to -8°

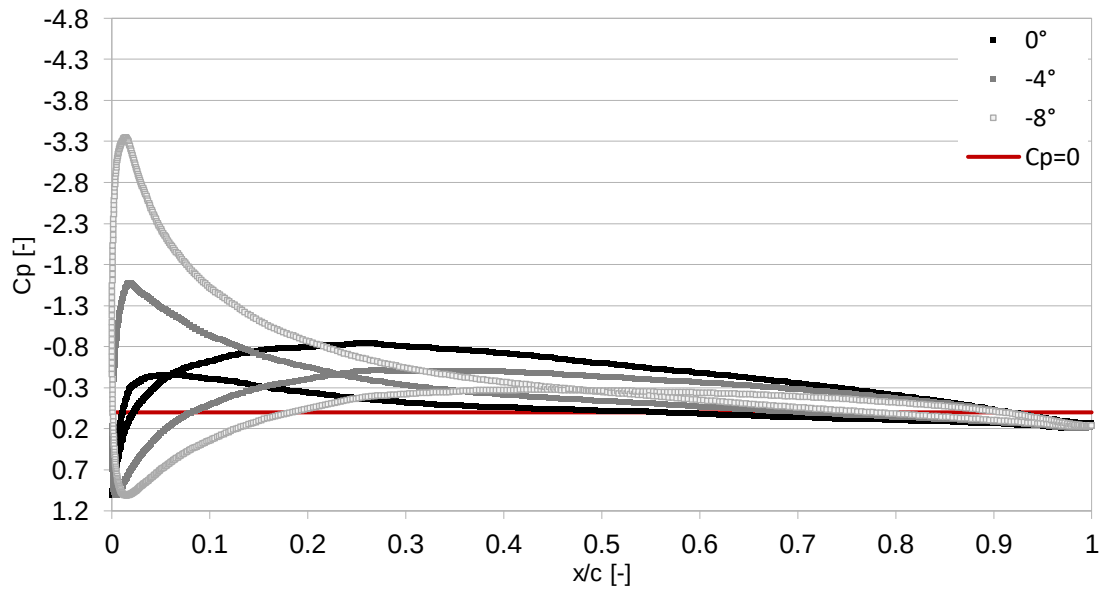


Figure 35: Pressure coefficient C_p for selected negative alphas

Figure 34 and Figure 35 show the pressure distributions for negative α . After a short positive section at the tip, both curves for upper and lower surface are on the negative side (see analysis).

The following Figures 36 to 40 show selected plots with pressure and velocity magnitudes which are also mentioned in section 5. The complete range of velocity and pressure plots is listed in a separate polybox folder.

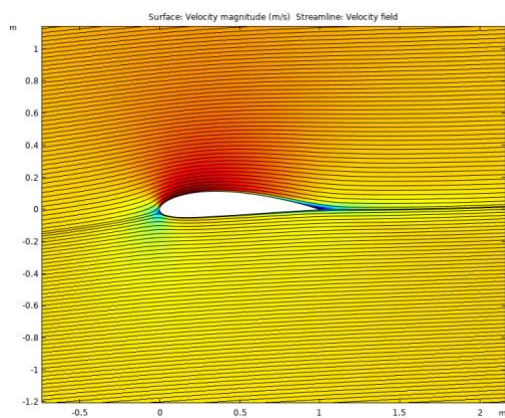


Figure 37: Velocity magnitude for $\alpha = 4^\circ$

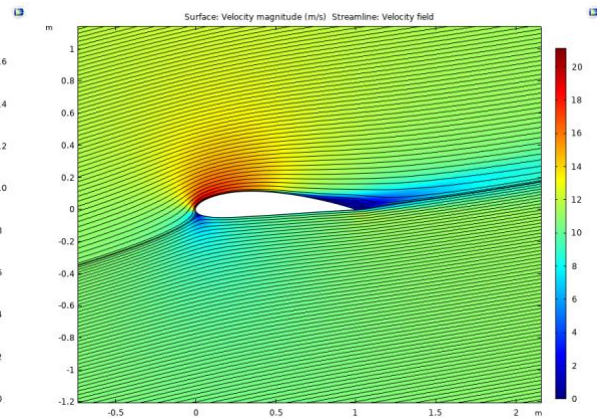
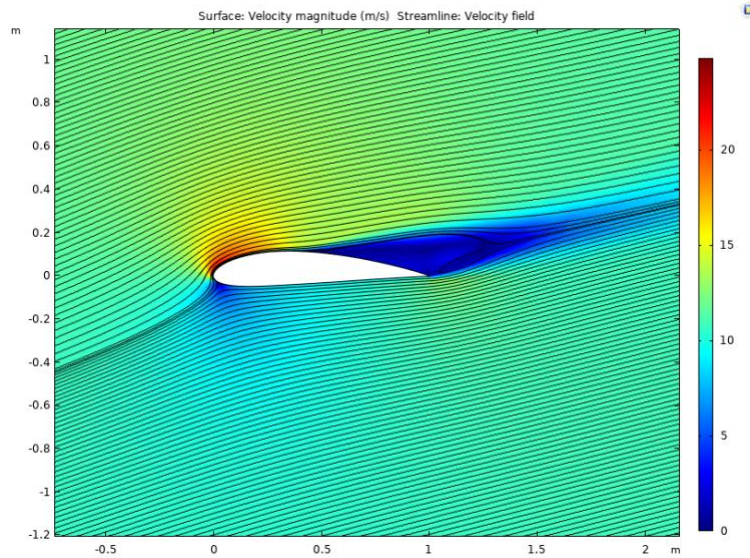
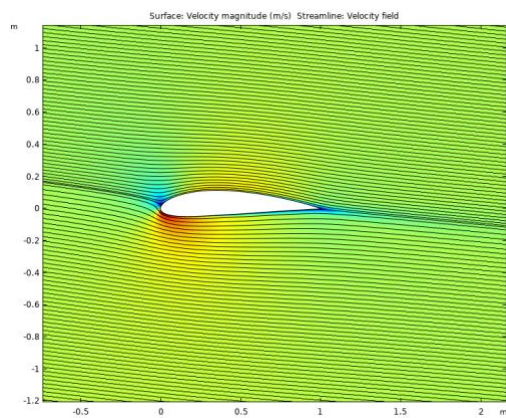
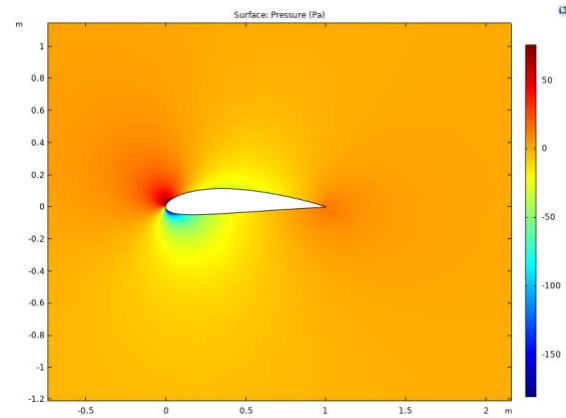


Figure 36: Velocity magnitude for $\alpha = 12^\circ$

Figure 40: Velocity magnitude for $\alpha = 16^\circ$ Figure 39: Velocity magnitude for $\alpha = -6^\circ$ Figure 38: Pressure magnitude for $\alpha = -6^\circ$

4.4.3 Variation of k_{inf}

As mentioned in section 3.4.6, the C_p distributions for a wide range of k_{inf} look indistinguishable. They are therefore not included in this section but listed in a separate Excel file.

4.4.4 Lift coefficient vs. angle of attack

As mentioned in section 3.4.7, the COMSOL model for NACA 0012 directly calculates C_l for a range of α . These values are then plotted in Figure 41.

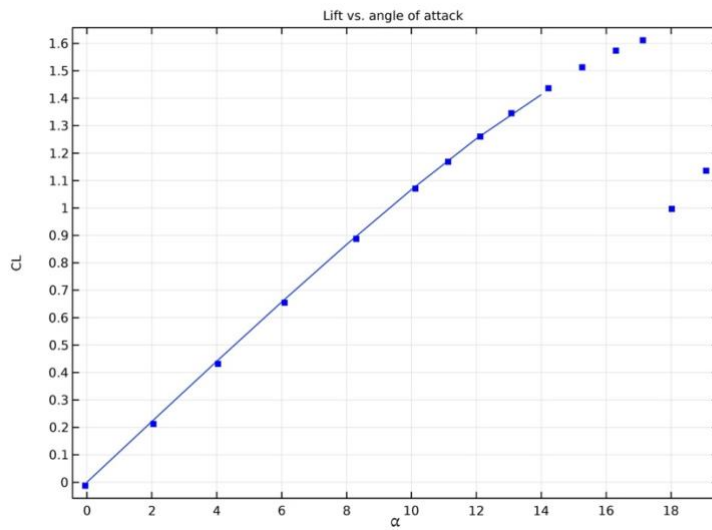


Figure 41: Computational (line) and experimental (dots) results for the lift coefficient vs. angle of attack

Similarly, the lift coefficient C_l is copied from the model to an Excel file for each angle of attack α and Figure 42 is created.

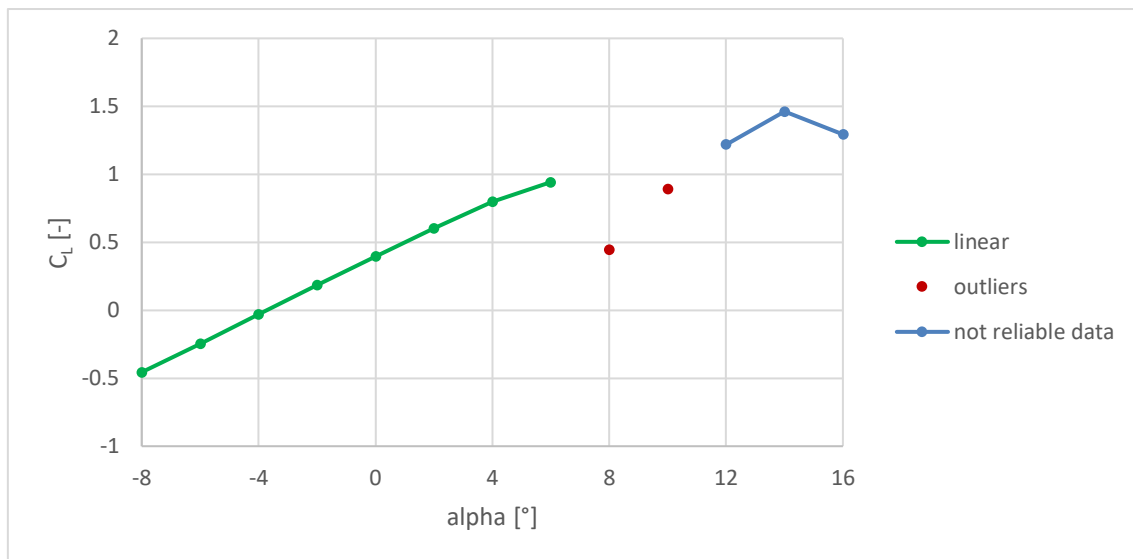


Figure 42: Lift coefficient C_l vs. angle of attack α

From $\alpha = -8^\circ$ to $\alpha = 6^\circ$ the relation between C_l and α is nearly linear. From 6° to 8° there is a significant drop of lift. The C_l for 8° and 10° were considered outliers due to the same reasons as those illustrated for the C_p plots. They were displayed accordingly. Due to the interruption in the graph, higher α s were also marked separately, and we must be careful when drawing conclusions in this range.

5 Analysis

5.1 Water flow in a simple tube

For this tutorial, as well as for the next one, we will talk about what was really important for our experience with COMSOL. Without this tutorial, it would certainly have been more difficult and complicated to start our project. Moreover, although extremely simple and predictable, the results allow us to make some interesting observations.

5.1.1 Representation of the velocity distribution

The plot of the velocity distribution shows how the velocity increases as the thickness of the tube decreases. Consequently, as soon as the liquid enters the last section with a larger area, it tends to decrease in velocity, becoming more homogeneous as it proceeds. Thanks to the hydraulics lessons taken during the third semester here at ETHZ, we have the possibility to give a simple and quick explanation to the values obtained, which in fact follow our expectations. The value of the velocity v is given by the ratio between the flow (Q) at a given instant and the area (A) of the section at that instant ($v = \frac{Q}{A}$). Thanks to the continuity equation (also known as Leonardo's equation), we also know that we will always observe the same value of Q inside our tube ($Q_i = Q_f$). At this point, the meaning of the velocity distribution becomes clear: the greater the value of A , the greater the denominator of Q , and the lower the value of v at that instant and in that place. On the other hand, narrower sections result in significantly higher speeds.

5.1.2 Representation of the pressure distribution

Let us now observe the graph of the pressure distribution inside the tube caused by the water flows. To analyze it, we will take advantage again of the hydraulics lesson, more precisely the Bernoulli equation. This equation states that in an ideal fluid (on which no work is applied), for every increase in the flow velocity there is simultaneously a decrease in pressure or a change in the potential energy of the fluid. Obviously, the opposite is also true (an increase in pressure is followed by a decrease in velocity).

The highest pressure is obtained at the meeting point of the two water flows (remember in fact that there are two "inlet" points, that is the upper end and the lower end of the rightmost rectangle). This result was certainly intuitive, as the collision of the flows causes obvious turbulence inside the pipe. Moreover, this certainly makes sense if we look at the Bernoulli's equation: inside the whole vertical section we observe very low velocities, which translate into high pressures.

Continuing, the liquid enters the second section, which is the first narrow horizontal rectangle. Here we will observe almost the opposite: the great increase in the flow

velocity (due to a decrease of the thickness) leads to a drastic decrease in the pressure values, which even reach values below zero (pressure lower than the atmospheric one).

In the end, the water reaches the last two horizontal sections, which have bigger surfaces. Here we see a gradual decrease in speed which, following Bernoulli, results in a gradual increase in pressure. The values will arrive close to zero and will tend to stabilize as we continue in the tube (if it had been longer, we would not have observed any particular peak/variation, because also the speed would have stabilized between 5 m/s and 8 m/s).

This first tutorial was certainly useful in many ways, great in our opinion for those of us who want to start their own projects with COMSOL.

5.2 2D and 3D tubes with complex geometry

Analyzing the two experiments, one aspect of COMSOL immediately stands out: the software allows modelling any desired shape in a fast and intuitive way. Whatever feature you want to add to your model (curvatures, measures, extrusions, thicknesses, ...), the software allows you to do it with a few "clicks", making the geometric construction of the desired body pleasant.

In addition, we quickly realised that several seemingly time-consuming steps (which we didn't perform in the smartest way), could be solved with very few steps thanks to simple tools. An evident example is the tool "mirror": not knowing such functions, the first times we have reproduced piece by piece what we needed, going then to reposition it in a symmetrical way.

This was of course the result of our poor experience, and it is precisely for this reason that the tutorials have been very helpful.

5.3 NACA 0012 airfoil

5.3.1 Representation of the velocity and the pressure distribution

We decided to comment on the velocity and pressure representations together without differentiating them, because they are extremely related to each other (as we have already seen for the first experiment).

In general, the highest velocities are observed at the locations with the most negative pressures. Moreover, it is easy to see that where there are no visible pressure gradients there are no particular changes in velocity.

At the points of collision perpendicular to the airflow and at the top of the tail, the pressure values are closest to zero, as already mentioned in the results section. Therefore, it is to be expected that even in the velocity representation we will find limit values as well as 0 m/s.

The concepts we have just discussed will be analyzed in more detail in the section on the Aventa Airfoil, which is why proposing a detailed discussion of these results seemed not necessary. The most important aspects are not only repeated, but also more interesting.

5.3.2 C_p plot

Regarding this plot in Figure 25, we can say that the trend along the perimeter follows exactly what we imagined and expected: the absolute maximum values of c_p are observed in the left side of the airfoil, as well as where the airflow hits the body. As we go further to the right, the absolute values tend to decrease in both curves, reaching the minimum value of 0.

It is also important to distinguish the two curves which we can do with the help of the pressure distribution representation. From it we understand that the lower part of the airfoil, where there are no great pressure changes, is represented by the curve below, which reaches the maximum value of 1. The upper surface where we find the greatest velocity is represented by the upper curve of the graph.

5.3.3 Lift-coefficient vs. angle of attack

Also in this section, we see the results we expected: Increasing the angle of attack also increases the Lift coefficient (the two values increase linearly together). This obviously makes sense, because the larger the α , the more the airflow will come "from below" and the more the airfoil will experience a force on the bottom.

It's interesting to note that for α equal to zero, we will get a Lift coefficient also equal to zero. This phenomenon is due to the symmetrical geometry of this airfoil and represents a fundamental factor for our final design (as we will see later in the report).

As already mentioned, we would like to point out that regarding this tutorial, we decided to do a rather superficial analysis. Going into detail would have been unnecessary and repetitive since most of the aspects will be analyzed in detail in the following section 5.4.

5.4 Aventa airfoil

Figure 27 shows that the C_p distribution obtained by the COMSOL model is very similar to the ones given by Xfoil and the other CFD. Generally, the COMSOL values lie in the middle of the two. We observed a noticeable bend at around $x/c = 0.6$ for Xfoil. Because we lacked insight into the Xfoil model, it is difficult to find out the reason for this bend, especially because it didn't occur in the other two simulations. It might result from a local area of turbulent flow that led to a slightly higher negative pressure.

The pressure is predominantly negative, except for one side on the second half and both sides right at the tip. With the vertical y/c values from the Xfoil or CFD distribution, the curve with the more negative pressure can be assigned to the upper surface of the airfoil. This is consistent with the pressure plot in Figure 28 that shows a low pressure throughout the entire upper surface and a less pronounced low-pressure area on the lower surface. Furthermore, the local high pressure on the tip is visible.

This pressure distribution also conforms with the actual function of an airfoil: to create a positive force that lifts the airfoil, to either generate energy or enable a plane to fly. The geometry is chosen in a way that the air travels longer on the upper side than on the lower side. The hereby resulting net negative pressure can be represented by a positive force pointing upwards. This force is called lift force and is further discussed in section 5.4.4.

The similarity can be attributed to the usage of the same $k-\omega$ SST turbulence model. The equations used to calculate the solution were therefore identical. The small deviations could have originated from different boundaries of the outer domain or a slightly different mesh. With the available knowledge, it was not possible to trace these deviations back to the actual reason.

This comparison of distributions was very crucial for the following experiments since it was a type of validation for the COMSOL model. It proved that the adaptation of the geometry and the creation of the new mesh worked out as planned and allowed us to proceed to changing the inflow conditions. One next interesting step would be to also vary the inflow conditions, namely the angle of attack α to find out where the limitations of the simulations are in comparison. We expect for higher α , to see greater deviations since the airfoils begins to stall and the uncertainty in the simulation is getting higher.

5.4.1 Variation of U_{inf}

Figure 29 shows that when U_{inf} is increased, the absolute value of C_p decreases. The curves look very similar and seem to be just multiplied by a scalar. Analyzing the slowest and fastest U_{inf} of 11.18 m/s and 16.18 m/s indeed shows that the C_p values always differ by a factor of 2.09. When comparing the maximum and minimum C_p values for each distribution, we see that the factor stays very similar for all the increments. It slightly decreases from 1.19 for both max. and min. between 11.18 m/s and 12.18 m/s to 1.14 between 15.18 m/s and 16.18. These multiplication factors are included in the separate Excel file for the U_{inf} -simulation.

The reduction of pressure for an increase of speed can be explained with Bernoulli's equation for steady flows (Wikipedia, 2021 A)

$$\frac{v^2}{2} + gz + \frac{p}{\rho} = \text{constant}$$

In the NACA 0012 tutorial the equation is obtained by solving the potential flow equation. First, the velocity potential φ is defined as $\mathbf{u} = -\nabla\varphi$ (COMSOL, 2021 B).

Next, this velocity potential must satisfy the continuity equation for incompressible flow, which is expressed as the following Laplace equation: $\nabla \cdot (-\nabla\varphi) = 0$ (COMSOL, 2021 B). This equation is called the potential flow equation.

Finally, the pressure is calculated with Bernoulli's equation for steady flows:

$$p = -\frac{\rho}{2} |\nabla\varphi|^2$$

This equation can only be used for two points on the same streamline. So, we can't for example use it with a point on the upper surface and one on the lower surface.

But we can say that as the airflow gets faster, the pressure decreases as stated in Bernoulli's equation.

5.4.2 Variation of α

Figure 32 shows that when α is increased, the absolute pressure on the upper surface is increased too. On the lower surface, however, the maximum C_p stays constant. This can be verified with Table 1, which shows the maximal and minimal C_p for each distribution. Similar to the U_{inf} -simulation, the multiplication factor between the maximal values is roughly constant at least for lower α 0° to 6°.

Table 1: Maximal and minimal C_p values for $\alpha = 0^\circ$ to $\alpha = 6^\circ$ and the ratio between these values

α	0°	2°	4°	6°
Maximal C_p	1.007	1.009	1.007	1.008
Minimal C_p	-0.844	-1.018	-1.237	-1.534
Max(i)/Max(i+1)	0.998	1.002	1.000	
Min(i)/Min(i+1)	0.829	0.823	0.807	
$i = 0^\circ, 2^\circ, 4^\circ$				

The pressure on the lower surface stays much longer positive (below the red line) for higher α , and for $\alpha = 12^\circ$ it even remains positive. The lift is therefore higher for a higher angle of attack. Due to the higher angle, the air travels even longer on the upper side, resulting in an even higher net negative pressure and thus lift force.

As mentioned in the section 4.4.2 and visible in Figure 31, the flow of air in the rear part of the airfoil is highly turbulent for $\alpha = 8^\circ$ and 10° . These were the angles of attack that at first had an error that didn't let the simulation terminate. Lowering the *number of iterations* removed the error but might have led to other complications that produced incorrect data. The convergence would definitely have to be checked with a *residuals convergence criteria* (see in Discussion).

Figure 30 also shows the deviating pressure for $\alpha = 8^\circ$ after around $x/c = 0.6$. The two angles of attack 8° and 10° were therefore not included in the general plot for C_p (Figure 32) which explains the larger gap between the curves.

Figure 37 shows that at $\alpha = 4^\circ$, the airstream is still attached to the airfoil along the whole upper surface. As expected, the velocity is higher on the upper side due to the longer distance along this streamline.

When α is increased, the airstream is slowly detaching from the upper surface on the second part of the airfoil. Very slow speed in this area as seen in Figure 36 is the direct consequence thereof. At $\alpha = 16^\circ$ (Figure 40) the airstream is even more detached and the blue area with no streamlines and low velocity is much bigger. We expect the airfoil to stall at around this angle of attack (see 5.4.4).

For negative α (Figure 34 and Figure 35), the curve with the strongly negative pressure on the tip now corresponds to the lower surface. This can be verified with Figure 38: The blue area (low pressure) is on the lower side and the red area (high pressure) is on the upper side. As with $\alpha = 0^\circ$, both curves in Figure 35 lie mostly on the negative side. Because the difference between the curves gets smaller with decreasing α , the lift force also decreases or even changes to a force pointing downwards. The negative lift coefficient C_l for α smaller than -4° supports this assumption.

5.4.3 Variation of k_{inf}

As a last parameter, the Free-Stream turbulent kinetic energy k_{inf} was varied by multiplying it with a wide range of factors. As already mentioned, the C_p values changed only minimally, and the change was not visible in the plots. This k_{inf} either really doesn't have an impact on the airflow and the pressure on the airfoil or the influence of the parameter wasn't completely included in the methods used for this simulation and should have been accounted for in a separate function.

5.4.4 Lift coefficient vs. angle of attack

Figure 42 shows the lift coefficient C_l vs. the angle of attack α . Compared to Figure 41 for the NACA 0012 tutorial, the same almost perfectly linear correlation between α

and C_l can be seen for an angle of attack up to 6° . Due to the symmetry in NACA 0012, the curve in Figure 41 starts at (0,0). The Aventa airfoil is asymmetrical in a way that the distance for the upper airflow is larger than the one for the lower airflow. This creates a positive lift coefficient even for the negative $\alpha = -2^\circ$. The C_l for 8° and 10° is unreliable due to the turbulent flow and possible error mentioned above. The values were therefore marked separately as outliers. Because of the large gap between 6° and 12° , the larger α s were marked in a different color. Based on the pressure and velocity plots, we expect the airfoil to stall at around 16° or 18° and the small drop between 14° and 16° in C_l could indicate such a stall. However, the data must be viewed and analyzed with caution. Moreover, the drag coefficient C_d must be taken into account. (Lyu, 2015) According to (Wikipedia, 2021 B) the general formula for C_d is

$$C_d = \frac{2F_d}{\rho u^2 A}$$

where:

F_d is the drag force in the direction of the flow velocity

ρ is the mass density of the fluid

u is the flow speed of the object relative to the fluid

A is the reference area

Unfortunately, we have not been able to implement this drag coefficient in the COMSOL model, among other things because we learned of the importance of this C_d quite late in the process.

6 Discussion

This report showed what could be achieved when using the software COMSOL and run several fluid-flow simulations on an airfoil of a wind turbine with having almost no experience in this field. To be honest, it was not as much as we initially had hoped or expected. COMSOL is an intuitive software with a graphic interface that makes first approaches and implementing a geometry and the underlying physics comparably easy. The NACA 0012 tutorial helped tremendously in analyzing the results by directly providing helpful tools such as the line integration to calculate the lift coefficient C_l or the various velocity and pressure plots that were easily adaptable to our requirements.

When it came to the numerous errors we encountered and most importantly analyzing the obtained results, things got much more difficult. Already adapting the model to the new geometry and, as a consequence, recreating the mesh proved to be a hurdle that took us several approaches and discussions with our supervisors.

Nonetheless, the results obtained were very interesting. Especially the experiment with the variation of the angle of attack α told us a lot about how fluid flows around a structure and which effect the geometry of the airfoil had on the airflow.

The shape of the airfoil was designed in a way that even for a negative angle of attack of -2° , the lift coefficient was still positive and thus the lift force pointing upwards. As α was increased, C_l also got bigger. One could conclude that the increase of α has no limit. Figure 41 and Figure 42 however show that this process has a limit. The airstream is getting more and more detached from the upper surface of the airfoil until the critical angle of attack is reached (Wikipedia, 2021 C). The airfoil then starts to stall which is indicated by the drop of lift coefficient C_l . This critical α is unique for each airfoil and typically in the range of 8° to 20° (Wikipedia, 2021 C). For the NACA 0012, it is between 17° and 18° (Figure 41). For the Aventa airfoil, we expect it to be around 14° and 16° but further simulations would have to be done to verify the exact value. Finding the α that creates a high lift coefficient is essential for using the airfoil more efficiently and therefore generating more energy.

The numerous errors, especially the ones for $\alpha = 8^\circ$ and 10° , showed us that our COMSOL model has its limitations. Finding where exactly the cause of this error lied proved to be very difficult. The mesh was adapted and some solver configurations were changed. This resolved some but not all the problems.

Hence, the first step of expanding the simulations would be to rigorously look into the errors and find a way that the results for $\alpha = 8^\circ$ and 10° become more reliable. After doing some research, we found out that *residuals convergence criteria* are a crucial part in every computational fluid dynamics simulation (CFD). For a solution to converge, the chosen residuals have to be below a certain number. For wind turbine blade modeling the typical value is 10^{-4} (Rajendra & Awadhesh, 2020).

Next, one would implement the fluid structure interaction (FSI) in the existing model. This could be done with COMSOL's integrated *fluid-solid interaction* models by adding new physics and specifying the turbulence model for example as a k- ω simulation. The solid mechanics would have to be defined as well as the *Fluid-Structure Interaction* under *Multiphysics*. We started the first steps and used the same initial values for the fluid properties as in the previous simulations. Very quickly, though, we realized that the FSI was even more complicated to set up than the previous model, where we had some guidance from the tutorial.

Mr. Abdallah kindly sent us a COMSOL model, where a moving mesh was chosen as an approach to simulate the FSI. We looked into the model and discovered familiar parts such as the section *Laminar Flow* with its settings as well as the very simple geometry. Unfortunately, we haven't had the time to adapt the ideas of this model to our model. We were focused on finishing the simulations we already had and most importantly extract as many results as possible. Nevertheless, we were disappointed, that we ultimately didn't reach this initial goal and looking back, we think more would have been possible, had we managed to finish the earlier simulations with less interruptions and difficulties.

Despite this, we are very grateful to have had the opportunity to learn that much in an entirely new field and that still some very interesting results could be received.

7 Bibliography

- Rajendra, R., & Awadhesh, K. (2020, September 7). Fluid-Structure Interaction of Wind Turbine Blade Using Four Different Materials: Numerical Investigation. *symmetry*, 1467(12(9)).
- COMSOL. (2021 B, June). *Tutorial: Flow Around an Inclined NACA 0012 Airfoil*. Retrieved June 2021, from COMSOL Tutorials: <https://www.comsol.ch/model/flow-around-an-inclined-naca-0012-airfoil-14629>
- C2ES, C. f. (2021, June). Retrieved from Renewable Energy: <https://www.c2es.org/content/renewable-energy/>
- COMSOL. (2017, July 16). *COMSOL Blog*. Retrieved from Which Turbulence Model Should I Choose for My CFD Application?: <https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/>
- COMSOL. (2021 A, June). *COMSOL doc*. Retrieved from The SST Turbulence Model: https://doc.comsol.com/5.5/doc/com.comsol.help.cfd/cfd_ug_fluidflow_single.06.091.html
- IRENA, I. R. (2021, June). Retrieved from Wind energy: <https://www.irena.org/wind>
- Jason M. Jonkman, M. L. (2005). *FAST User's Guide*. National Renewable Energy Laboratory 1617 Cole Boulevard, Golden, Colorado.
- Lyu, P. (2015, June 16). *COMSOL Blog*. Retrieved from How Do I Compute Lift and Drag?: <https://www.comsol.com/blogs/how-do-i-compute-lift-and-drag/>
- SIMSCALE. (2021, May). *SimScale Documentation*. Retrieved from K-Omega and K-Omega SST: <https://www.simscale.com/docs/simulation-setup/global-settings/k-omega-sst/>
- The Defeated Engineer. (2020 A, July 14). *YouTube*. Retrieved from How to use a Polygon to build a Complex Geometry | Tutorial #2: <https://youtu.be/IlmjkvupzaA>
- The Defeated Engineer. (2020 B, October 3). *YouTube*. Retrieved from Creating Junction using Cylinder & Torus | COMSOL | Tutorial #3: <https://youtu.be/-SIJOU4s4rU>
- The Defeated Engineer. (2020 C, July 14). *YouTube*. Retrieved from Getting Started with COMSOL Multiphysics | Tutorial #1: <https://www.youtube.com/watch?v=1-ov7c-L7Wc>
- The Editors of Encyclopaedia Britannica. (2019, May 30). *Windmill*. Retrieved from The Editors of Encyclopaedia Britannica: <https://www.britannica.com/technology/windmill>
- Wang, L., Quant, R., & Kolios, A. (2016, November). Fluid structure interaction modelling of horizontal-axis wind turbine blades based on CFD and FEA. *Journal of Wind Engineering and Industrial Aerodynamics*, 158, 11-25.
- Wikipedia. (2021 A, June). Retrieved from Bernoulli's principle: https://en.wikipedia.org/wiki/Bernoulli's_principle
- Wikipedia. (2021 B, June). Retrieved from Drag coefficient: https://en.wikipedia.org/wiki/Drag_coefficient
- Wikipedia. (2021 C, June). Retrieved from Stall (fluid dynamics): [https://en.wikipedia.org/wiki/Stall_\(fluid_dynamics\)](https://en.wikipedia.org/wiki/Stall_(fluid_dynamics))

8 Appendix

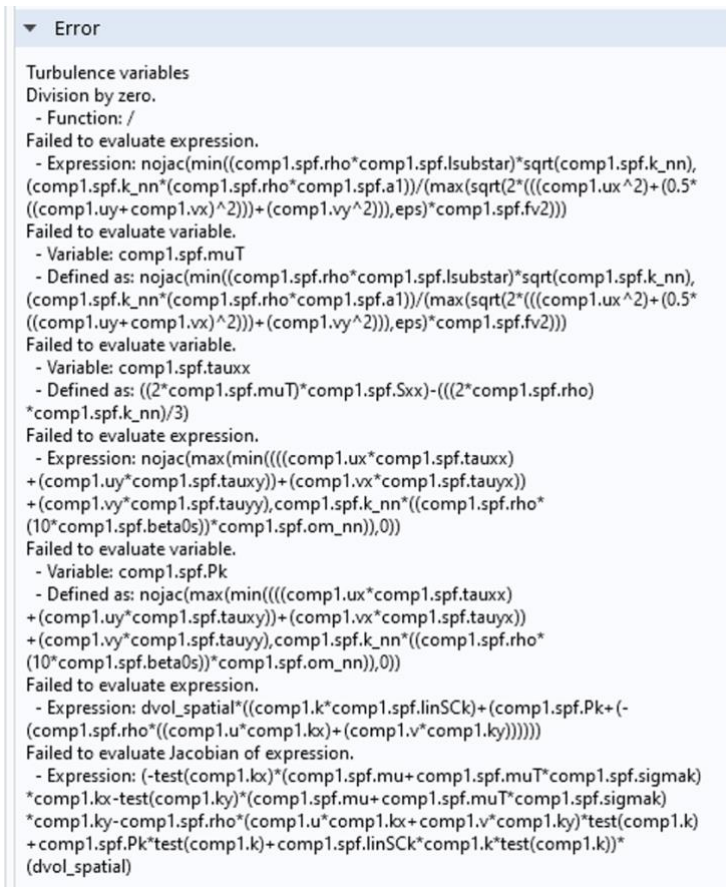


Figure 43: Initial error for $\alpha = 8^\circ$ and 10°

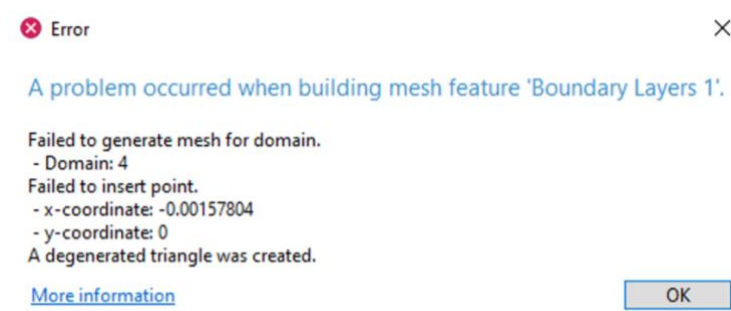


Figure 44: Error in the first attempt of creating the mesh

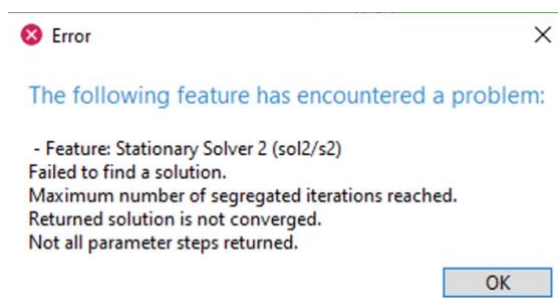


Figure 45: Small error that came up in higher alphas, but we were still able to export the results



Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich

Declaration of originality

The signed declaration of originality is a component of every semester paper, Bachelor's thesis, Master's thesis and any other degree paper undertaken during the course of studies, including the respective electronic versions.

Lecturers may also require a declaration of originality for other written papers compiled for their courses.

I hereby confirm that I am the sole author of the written work here enclosed and that I have compiled it in my own words. Parts excepted are corrections of form and content by the supervisor.

Title of work (in block letters):

Simulating the fluid-structure interaction of a wind turbine blade in COMSOL

Authored by (in block letters):

For papers written by groups the names of all authors are required.

Name(s):

Nydegger

Bruschetti

First name(s):

Josija

Fabrizio

With my signature I confirm that

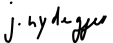
- I have committed none of the forms of plagiarism described in the '[Citation etiquette](#)' information sheet.
- I have documented all methods, data and processes truthfully.
- I have not manipulated any data.
- I have mentioned all persons who were significant facilitators of the work.

I am aware that the work may be screened electronically for plagiarism.

Place, date

Zürich, 23 June 2021

Signature(s)




For papers written by groups the names of all authors are required. Their signatures collectively guarantee the entire content of the written paper.