

# LIFT AND DRAG COEFFICIENT ANALYSIS IN A FOILING CENTREBOARD OF A MOTH WITH FLAP'S ANGLE OPTIMIZATION

Student: Alessandro Chiappalone  
 Course: Advanced Fluid Dynamics  
 Professor: Jan Pralits  
 OpenFoam Mentor: Matteo Colli

## ABSTRACT

This paper describes the basic of a foiling centreboard. This kind of layout is one of the most promising solution for sailboats. The goal of this work is to find a relation between flap's angle and drag/lift coefficient for a centreboard based on the GARDA TOP 3 geometry. The study was carried out using the software OpenFOAM.

## 1. Introduction

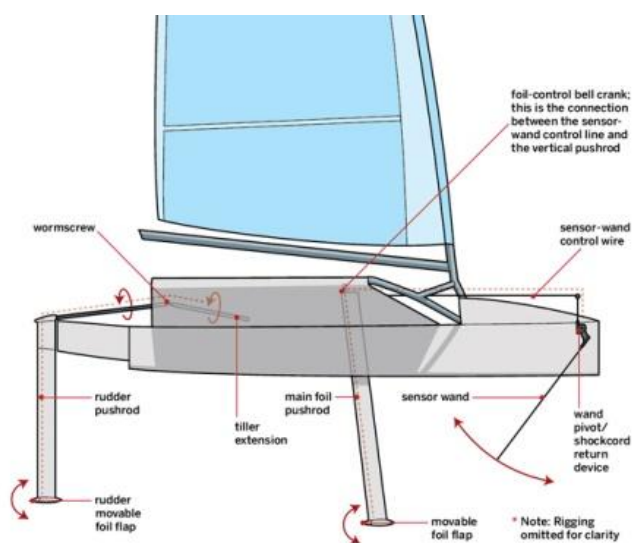
This work is based on a centreboard of a foiling moth (fig. 1.1), this kind of sailboat is a craft to get a speed that would be impossible to achieve with a classic sailboat.

This is essentially due to the possibility given by foils to lift the boat over the sea level cancelling the drag between hull and sea surface. The overall weight is really low, around 30 kg.

The combination of this factors allows to reach the speed of 30 knots.

The foiling moth is really complicated because the flap's angle regulation is function of the craft speed and the realization of a model that could improve all variables with a cfd program like OpenFoam would be difficult.

Figure 1.1 : a foiling moth



For this reason the study was based only on the centreboard.

The aim of this work is to find (for the first step) the value of drag ( $C_D$ ) and lift ( $C_L$ ) coefficient of the centreboard. As second step to optimize the flap's angle to find the value that realize the best compromise between  $C_D$  and  $C_L$ .

## 2. Method

### 2.1 Setting the Problem

In order to choose the correct solver in OpenFOAM the Reynolds number (Eq. 2.1) was evaluated.

Eq. 2.1

$$Re = \frac{c L}{\nu}$$

We use as a characteristic length (L) the value of the chord of the centreboard.

The fluid's velocity (boat speed) chosen for the study is 5 m/s because this is a typical starting speed for this kind of craft.

The characteristic values used and the corresponding Reynolds numbers are found in Table 2.1

Table 2.1: Reynolds parameter

Variable	Mean	Unit	Value
$c$	Water Velocity	[m/s]	5
$L$	Characteristic Length	[m]	0,11
$\nu$	Kinematic Viscosity	[m <sup>2</sup> /s]	1.2 10 <sup>-6</sup>
<b>Re</b>	/	/	<b>458332</b>

### 2.2 Numerical Solver

OpenFOAM is an opensource software which solves the Navier-Stokes equations in terms of the pressure (P) and velocity (c) in each point of the mesh. The derivation of the N-S equations begins with an application of Newton's second law: conservation of momentum (often alongside

mass and energy conservation) being written for an arbitrary portion of the fluid.

In our case we deal with water and so we must use the incompressible form for the Navier – Stokes equations. Moreover, considering constant viscosity N-S equations are simplified:

Eq. 2.2

$$\rho \left( \frac{\partial c}{\partial t} + c \cdot \nabla c \right) = -\nabla p + \mu \nabla^2 c + f$$

$$\nabla \cdot c = 0$$

Table 2.2: N-S parameters

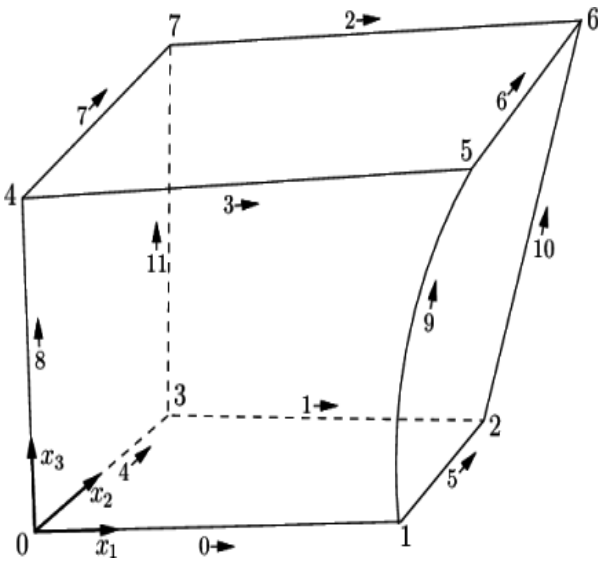
Variable	Mean	Unit
$c$	Water Velocity	[m/s]
$\rho$	Water Density	[kg/m <sup>3</sup> ]
$\mu$	Dynamic Viscosity	[Kg/ms]
$f$	Body Forces per Unit Volume	[N/m <sup>3</sup> ]

The first step in OpenFOAM is to create a control volume.

To resolve the Navier-Stokes equation it is necessary to fix the boundary conditions. These conditions are imposed on the surfaces of the control volume and they regards pressure and velocity. To understand the B.C. It's necessary to define the correct name that has been used for each surfaces of fluid domain.

To recognise surfaces OpenFOAM use the notation for vertices represented in Fig. 2.1.

Figure 2.1: a single cell block



Each plane can be expressed by using 4 numbers which are for example for the bottom plane [0 1 2 3].

In our case we have the following names:

```
boundary
(
  Inlet
  {
    type patch;
    faces
    (
      (1 2 6 5)
    );
  }
  Outlet
  {
    type patch;
    faces
    (
      (0 3 7 4)
    );
  }
  Sea
  {
    type patch;
    faces
    (
      (0 1 5 4)
      (0 1 2 3)
      (3 2 6 7)
      //(4 5 6 7)
    );
  }
  Ship
  {
    type wall;
    faces
    (
      (4 5 6 7)
    );
  }
);
```

The fluid domain is represented by a parallelepiped with an X side of 12 [m] an Y side of 2.4 [m] and a Z side of 2.5 [m].

X is the streamwise direction, Z axe is the lift direction (vertical).

Considering that, the boundary conditions are:

- Velocity:

```
dimensions [0 1 -1 0 0 0];
internalField uniform (-5 0 0);
boundaryField
{
  Inlet
  {
    type fixedValue;
    value uniform (-5 0 0);
  }
  Outlet
  {
    type zeroGradient;
  }
  Sea
  {
    type slip;
  }
  Ship
  {
    type fixedValue;
    value uniform (0 0 0);
  }
  Centerboard
  {
    type fixedValue;
    value uniform (0 0 0);
  }
}
```

- Pressure:

```

dimensions      [0 2 -2 0 0 0];
internalField   uniform 0;

boundaryField
{
  Inlet
  {
    type        zeroGradient;
  }
  Outlet
  {
    type        fixedValue;
    value       uniform 0;
  }
  Sea
  {
    type        slip;
  }

  Ship
  {
    type        zeroGradient;
  }
  Centerboard
  {
    type        zeroGradient;
  }
}

```

About B-C:

Condition	Mean
<i>fixedValue</i>	The value of magnitude is fixed and constant
<i>uniform</i>	In all points of the plane (inlet) the value is the same
<i>zeroGradient</i>	$\nabla \mathbf{c} = 0$ , $\nabla p = 0$
<i>slip</i>	Tangent velocity component on the wall is different from 0

For this kind of problem the solver that we have chosen was simpleFoam. SimpleFoam is an OpenFOAM solver used to solve steady incompressible turbulent flow based on a RANS analysis. The following equations are the incompressible Reynolds-Averaged Navier–Stokes equations (RANS).

Eq. 2.3

$$\nabla \cdot (\bar{\mathbf{c}}) = 0$$

Eq. 2.4

$$\frac{\partial \bar{\mathbf{c}}}{\partial t} + \nabla \cdot (\bar{\mathbf{c}}\bar{\mathbf{c}}) = \frac{-\nabla \bar{p}}{\rho} + \nu \nabla^2 \bar{\mathbf{c}} - \frac{1}{\rho} \nabla \cdot \tau^R$$

Where  $\tau^R$  is the Reynolds stress tensor and is given by:

Eq. 2.5

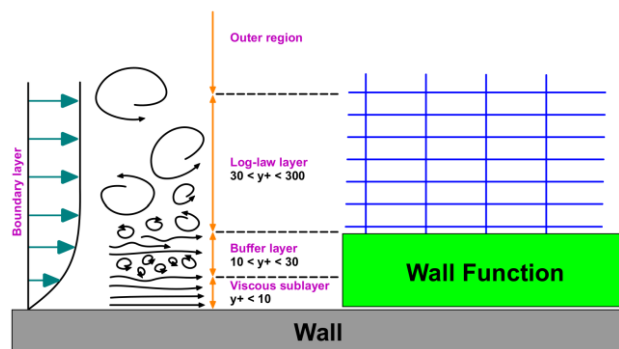
$$\tau^R = -\rho(\overline{\mathbf{c}'\mathbf{c}'})$$

There are some different turbulence models:

- $k - \epsilon$ : Robust. Widely used despite the known limitations of the model. Performs poorly for complex flows involving severe pressure gradient, separation, strong streamline curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies.
- $k-\omega$ : Superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows compared to models from the  $k$ -epsilon family. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbomachinery).

In Modern RANS solvers included in CFD packages such as OpenFOAM allow the user to apply an intermediate method that concentrates the main advantages of the  $k-\omega$  and  $k-\epsilon$  two-equation models, the SST  $k-\omega$  1 model (Menter [1993]). The use of a SST  $k-\omega$  method in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer, the formulation also switches to a  $k-\epsilon$  behavior in the free-stream and thereby avoids the common  $k-\omega$  problem that the model is too sensitive to the inlet free-stream turbulence properties. In our study we don't resolve directly the boundary layer but we use the "law-of-the-wall" to avoid too much refined mesh. The law of the wall consist using a wall function for the  $v_T, k, \omega$  variables close to surfaces.

Figure 2.4: representation of a wall function



In our case we use wall function because we are more interested in the phenomenon in the outer region, rather than the forces on the wall. Using wall functions with SST  $k-\omega$  model we have to obtain a  $y^+$  (Chapter 5.2) between 10 and 300 over surfaces like Ship and Centerboard. To have a good simulation is necessary to impose some boundary conditions of  $k, \omega$  and  $v_T$  (turbulent viscosity), where  $k$  and  $\omega$  are respectively the turbulent kinetic energy and the specific kinetic energy.

When the RANS model is executed  $k$ ,  $\omega$  and  $v_T$  follow the corresponding wall function (the values vary with the normal direction and remains constant along the direction tangential to the wall).

Below are reported the boundaries conditions for  $v_T$ ,  $k$  and  $\omega$ .

- $v_T$ :

```
boundaryField
{
  Inlet
  {
    type          calculated;
    value         uniform 0;
  }
  Outlet
  {
    type          calculated;
    value         uniform 0;
  }
  Sea
  {
    type          slip;
  }
  Ship
  {
    type          nutkWallFunction;
    value         uniform 0;
  }
  Centerboard
  {
    type          nutkWallFunction;
    value         uniform 0;
  }
}
```

- $k$ :

```
boundaryField
{
  Inlet
  {
    type          fixedValue;
    value         uniform 0.24;
  }
  Outlet
  {
    type          inletOutlet;
    inletValue    uniform 0.24;
    value         uniform 0.24;
  }
  Sea
  {
    type          slip;
  }
  Ship
  {
    type          kqRWallFunction;
    value         uniform 0.24;
  }
  Centerboard
  {
    type          kqRWallFunction;
    value         uniform 0.24;
  }
}
```

- $\omega$ :

```
boundaryField
{
  Inlet
  {
    type          fixedValue;
    value         uniform 1.78;
  }
  Outlet
  {
    type          inletOutlet;
    inletValue    uniform 1.78;
    value         uniform 1.78;
  }
  Sea
  {
    type          slip;
  }
  Ship
  {
    type          omegaWallFunction;
    value         uniform 1.78;
  }
  Centerboard
  {
    type          omegaWallFunction;
    value         uniform 1.78;
  }
}
```

### 3. Geometry

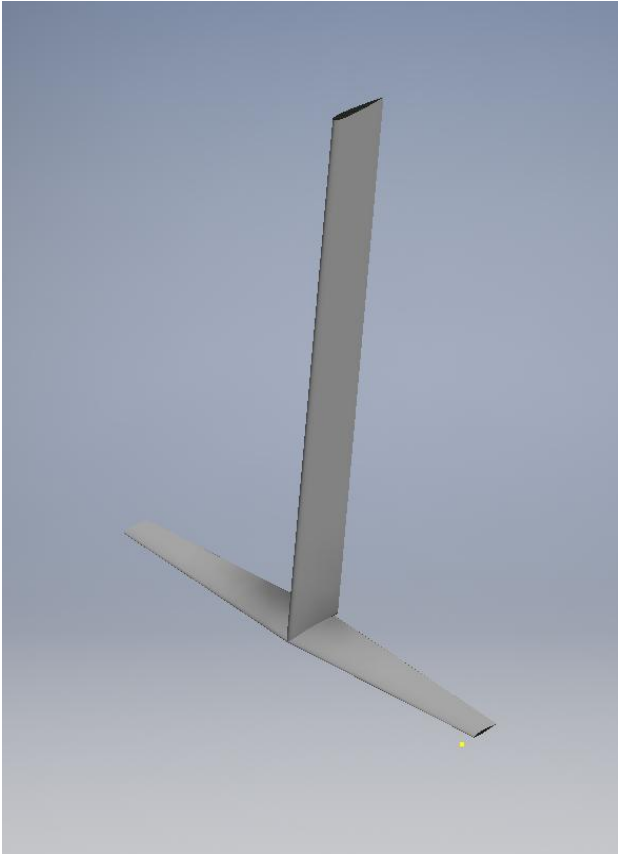
The centreboard geometry is based on the GARDA TOP 3 moth. There are some different type of foiling moth which have other performances and shapes. However using a GARDA TOP 3 we have a starting point to reach our goal. At the bottom of the centreboard there is an 860mm span asymmetric foil section that is set at approximately zero degrees to the bottom of the hull. The current foil section used for the lifting foils is a NACA 63412. Both the centreboard, and rudder hydrofoils have trailing edge flaps to allow the modification of their shape to influence the amount of lift produced.

The cad model was realised using the Inventor software and after imported on OpenFoam.

Table 3.1: dimensions and profiles

	Section	Span [mm]	Chord[mm]
Centreboard	NACA 0012	1000	120
Centreboard Foil	NACA 63412	860	110

Figure 3.1 : centreboard's cad for a 0° of trailing edge flaps



In this type of craft it is necessary to find the correct angle of lifting section relative to the hull. To make this we must consider not only the centreboard but also the rudder.

We have two different case that can give some issues:

Figure 3.2

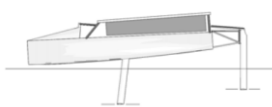


Figure 3.3

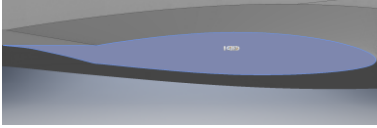
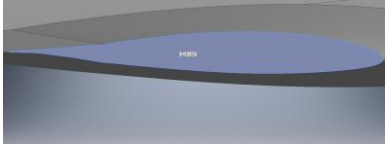
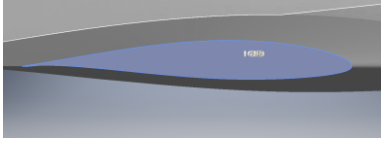
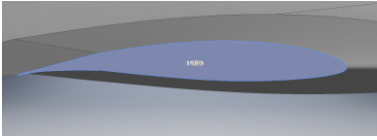
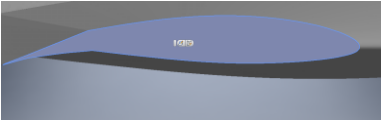


The figure (3.2) shows the case where the main foil is actually angled down slightly. With the flap deflected down, and due to the camber in the section the foil can still create lift when angled down, but a considerable bow up attitude would be required to get this at the right angle to produce sufficient lift. You'd have considerable transom drag in trying though, and foiling would be unlikely, or need plenty of wind. The image (3.3) shows the case where the main foil is positioned at a greater angle of attack, and so creates a lot of lift. This is likely to lift off quite easily but likely to create too much as even with the flap all the way up the foil would still be producing lift in the normal trim mode. To reduce the amount of lift the boat would have to be trimmed quite nose down, and balanced by raking the rudder underneath as well. (Foiling Guide)

To avoid problems shown in figure 3.4-3.5 we decided to assume a 7° rudder angle and consequently an 8° angle between the centerboard and the hull.

The study was carried out using 5 different angles for the trailing edge flaps with only one leading edge angle of 0 degrees:

Table 3.2: angles and C.A.D.

<i>Trailing edge flap's angle</i>	<i>CAD Model</i>
<b>-10°</b>	
<b>-5°</b>	
<b>0°</b>	
<b>5°</b>	
<b>10°</b>	

## 4 Meshing

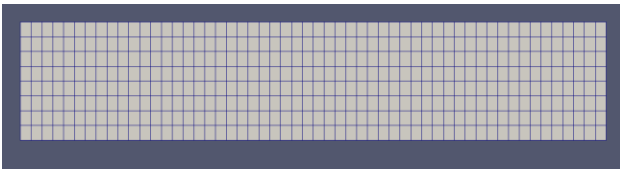
One of the most essential step of this work is the mesh and grid generation. After importing the geometry in OpenFOAM, it's important to customise the grid around it appropriately. For this purpose, OpenFOAM provides the right tools: blockMesh and SnappyHexMesh. SnappyHexMesh is a mesh generation utility for 3-dimensional meshes containing hexahedra and split-hexahedra automatically from triangulated surface geometries in Stereolithography format. The mesh approximately conforms to the surface by iteratively refining a starting mesh and morphing the resulting split-hex mesh to the surface. The specification of mesh refinement level is very flexible and the surface handling is robust with a pre-specified final mesh quality. Before snappyHexMesh is executed a background mesh this has

been done using BlockMesh. As shown in chapter 2.2 the fluid domain was realized by using BlockMesh command. We set its parameters to obtain a cell division similar as possible to cubes. Considering control volume dimensions we decided to divided it in this way:

$$[54 \ 8 \ 8]$$

So along X dimension we obtain 54 slice, 8 along Y direction and 8 along Z direction. The results was:

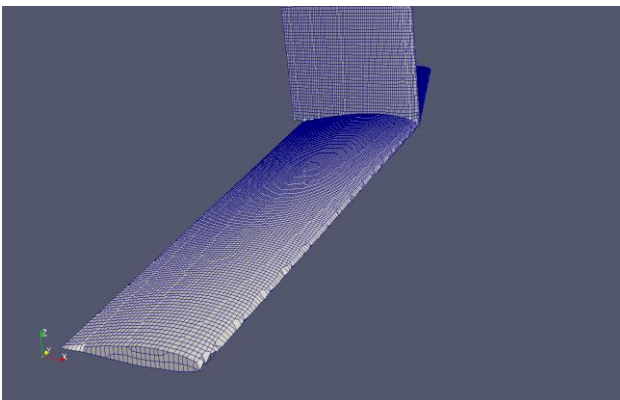
Figure 4.1 fluid domain mesh (with the same dimensions of figure 2.2)



The geometry was implemented in fluid domain using snappyHexMesh command. The meshing process was carried out in three different phases:

- 1) In a first time we started with the research of the best compromise between refinement surfaces edges and the calculator's power.

Figure 4.2: mesh on centerboard



- 2) As second step to achieve the correct value of  $y^+$  (chapter 5.2) has been useful increase the refinement level under the ship and close to the centerboard. To obtain this we used the addLayer tool that consent to have a better resolution near wall surfaces .

Figure 4.3: mesh around a foil's profile, showing layers close to shape for a profile with  $0^\circ$  .

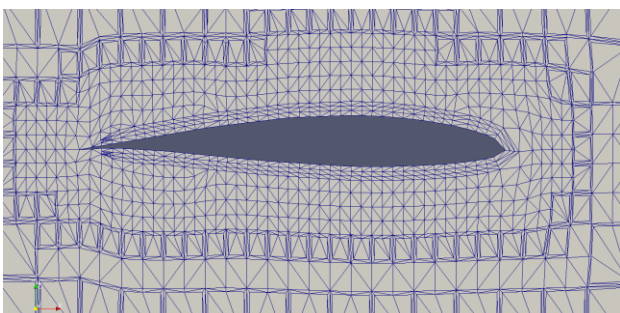
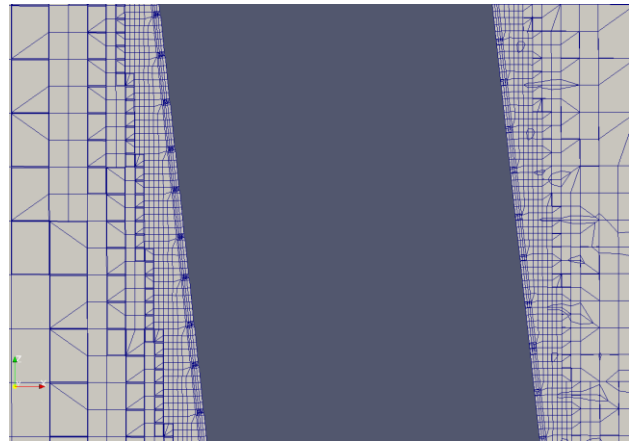
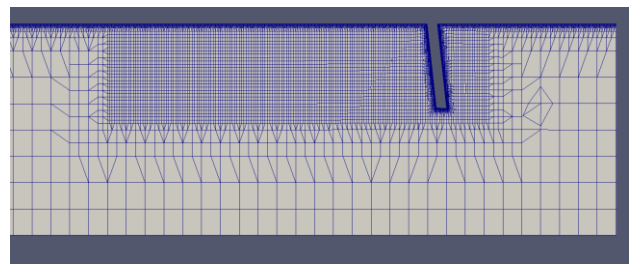


Figure 4.4: a particular of the centerboard showing layers close to the shape.



- 3) The last one step was increased the refinement beside the centerboard where there is turbulence. To achieve this has been created a box with snappyHexMesh dictionary close to geometry.

Figure 4.5: box beside centerboard



## 5 Results

### 5.1 Convergence study

Numerical simulations always need a grid convergence study. The convergence is guaranteed when there is an independence between results and mesh refinement (cells dimensions) .

To determinate the correct number of cells we made different simulations with different refinement levels until achieving a constant solution ( $C_L$  and  $C_D$ ).

This study was done only on the geometry with  $0^\circ$  for trailing edge flaps but it's applicable also on the other geometries. In table (5.1) are reported all 5 cases done to achieve the convergence:

Table 5.1 : convergence study cases

Cell's number	$C_L$	$C_D$
136000	1,0802	0,384936
643000	0,84637	0,248135
1550000	0,849123	0,269452
2550000	0,860901	0,279953
3740000	0,859068	0,276287

This refinement increment has been done around the centerboard, under the ‘Ship’ and beside the centerboard to have different resolution in the wake zone.

The results of the convergence’s study is reported in the Figure 5.1.

Figure 5.1 : lift and drag coefficient trends with cells number

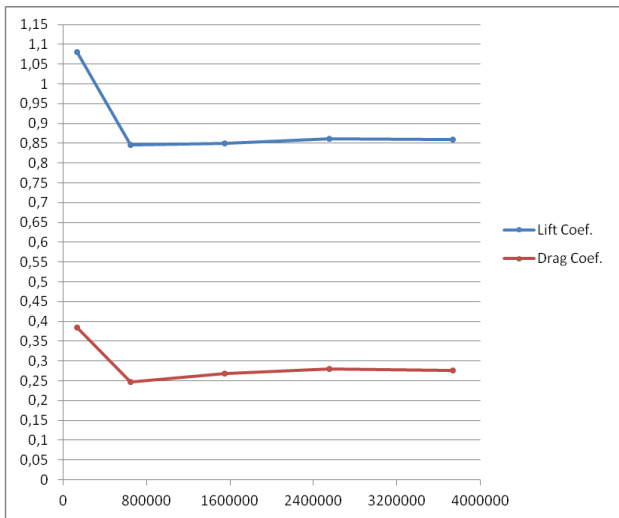


Figure 5.1 shows the trend of the coefficients. It’s possible to see that after 2500000 cells there is a very little difference between last two values.

To have a good resolution it was decided to work with the last number of cells reported in table 5.1.

Also other geometries have been solved with the same quantity of cells to achieve the same precision for each cases.

### 5.2 yPlus

The value called yPlus is a non dimensional quantity calculated close to wall. Wall unit function  $y^+$ , as defined in eq. 5.1, can constitute a guidance in selecting the appropriate grid spacing in the regions close to the surfaces.

Eq. 5.1

$$y^+ = \frac{y \cdot \mu_a}{\nu_a}$$

Eq. 5.2

$$u_\tau = \sqrt{\frac{\tau_w}{\rho_a}}$$

Eq. 5.3

$$\tau_w = \mu_a \left| \frac{\partial u}{\partial y} \right|_{y=0}$$

Where  $y$  is the distance to the wall,  $u_\tau$  is the friction velocity,  $\nu_a$  is the kinematic viscosity of the carrying fluid,  $\mu_a$  its dynamic viscosity and  $\tau_w$  the shear stress on the surface. In this particular case yPlus is determined on the walls called Ship and centerboard. To have a good simulation this value must to be between 10 and 300. This range of values is due to the fact that we use the law

of the wall (Chapter 2.2) to model the velocity profile close to wall instead of resolving it. To achieve this target it is necessary to work on the mesh and in particular on the layers near the walls. After a lot of tries, in table 5.2 are reported the finals yPlus values close to ship and centerboard:

Table 5.2: yPlus values

Trailing edge flap’s angle	Centerboard’s yPlus max	Ship’s yPlus max
-10	295	186
-5	280	150
0	175	165
5	255	145
10	260	165

With paraFoam it is possible to see the yPlus distribution and understand in which points the value is larger than in others .

Figure 5.1: yPlus distribution on a central centerboard section

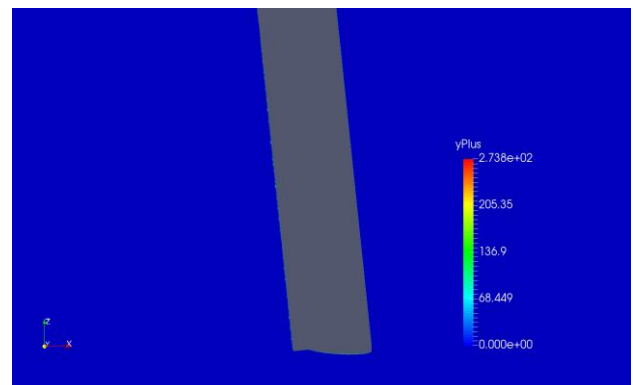
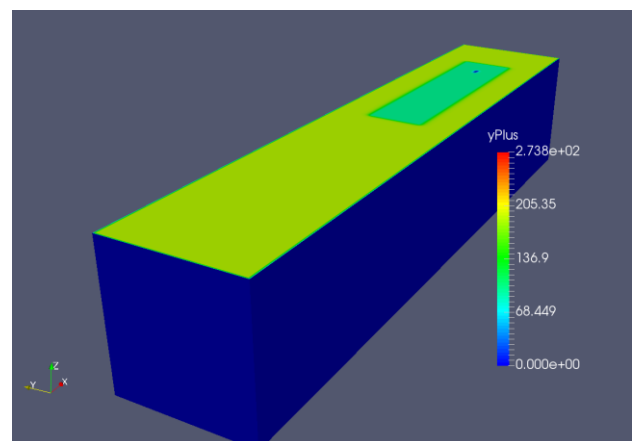


Figure 5.2 : yPlus distribution on the wall ‘Ship’.



### 5.3 Residuals and Time StepContinuity

Residuals and Time StepContinuity parameters represents the error that is committed on a determinate parameter, like Pressure and Velocity.

Residuals have to be much lower as possible to obtain a good simulation. Generally the residuals should be lower than  $10^{-4} - 10^{-5}$ . It's also important to monitor the residual's trend along the time. This is essential if we would like to know if our solution has become invariant with time.

Residuals go to convergence around 10 seconds in a total simulation of 15 seconds and their values are lower than  $10^{-5}$ . Other geometry have the same trend with the only differences about convergence time, indeed when flaps are used there is more turbulence and consequently residuals go to convergence later, around 12 s. Time step continuity represent the mass lose trough cells. To respect the mass conservation law it is essential that this value is lower as possible. Usually a number lower than  $10^{-11} - 10^{-12}$  is acceptable.

#### 5.4 Pressure (P) and Velocity (U)

Pressure and Velocity were calculated by OpenFoam in each cell center of the finite volumes grid and for each geometry. To have a general ideal of the results in figure 5.5 is showed the velocity and pressure results on the centerboard and on a section passing from centerboard's vertical axe.

Figure 5.5: pressure distribution on centerboard shape (case 0°)

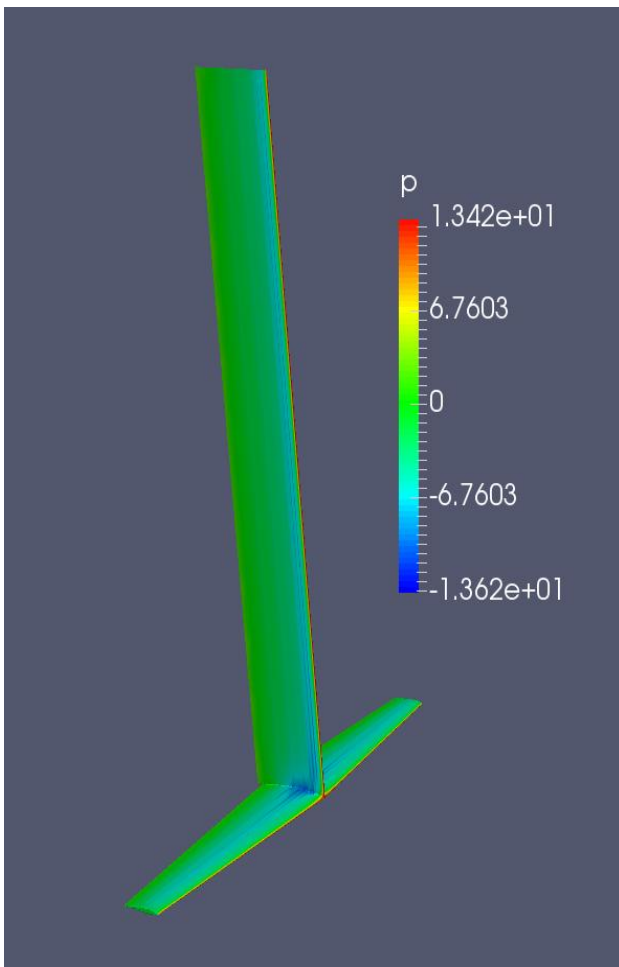


Figure 5.6: velocity distribution along fluid domain in a central section

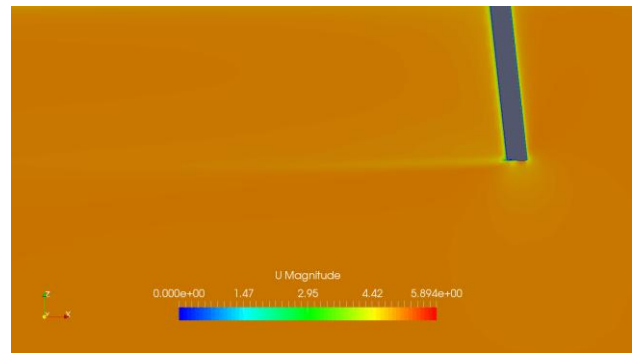
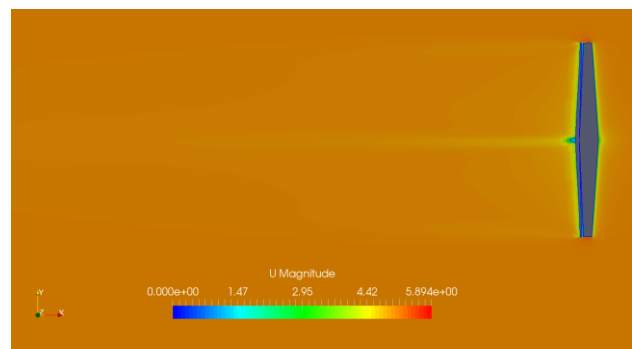


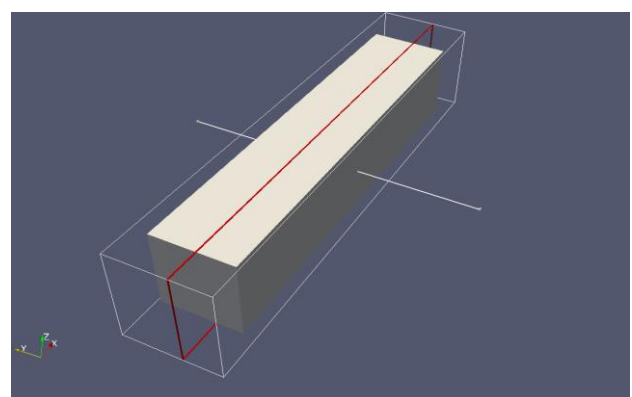
Figure 5.7: wake beside centerboard along a section normal to plane Z



In figures 5.6 and 5.7 it is possible to see a slight wake of turbulence beside the centerboard represented by the zone with orange shaded color.

To have a better description of the phenomenon it's important to see the pressure and velocity distribution around the foils profiles for each geometries. To make that has been cut the fluid domain along the plane showed in picture 5.8.

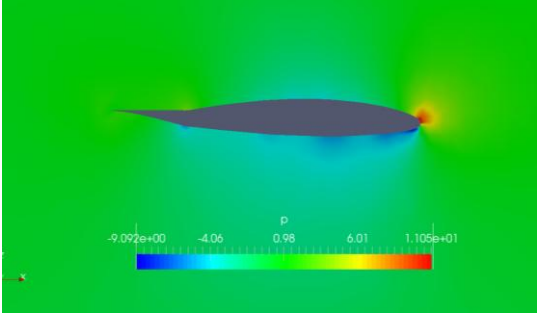
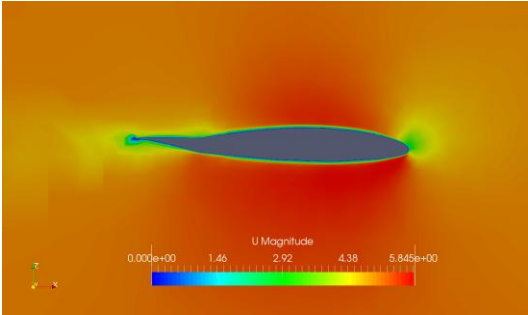
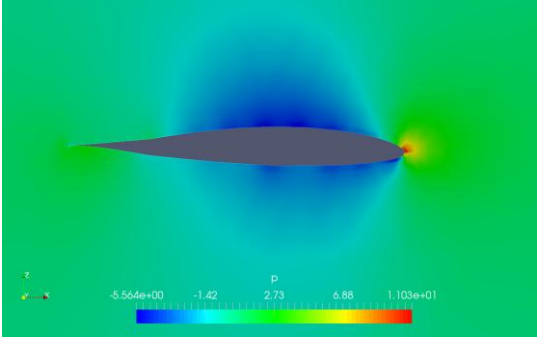
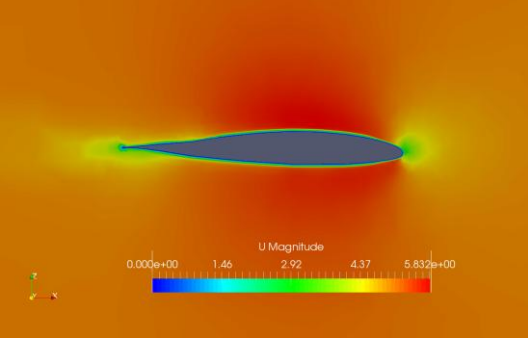
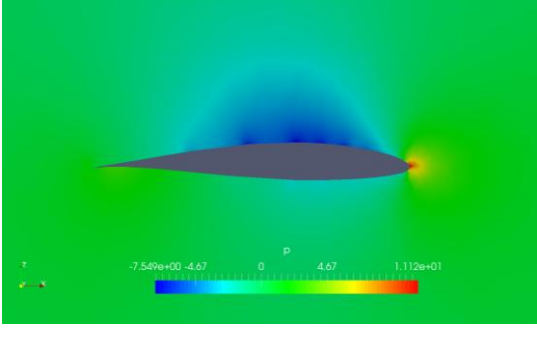
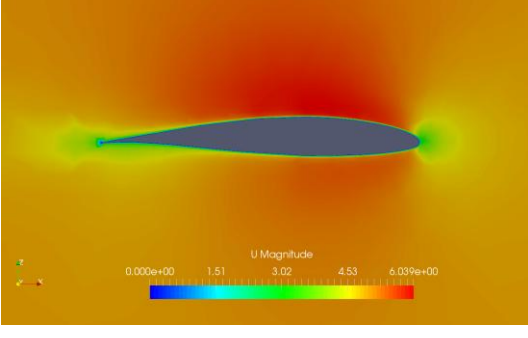
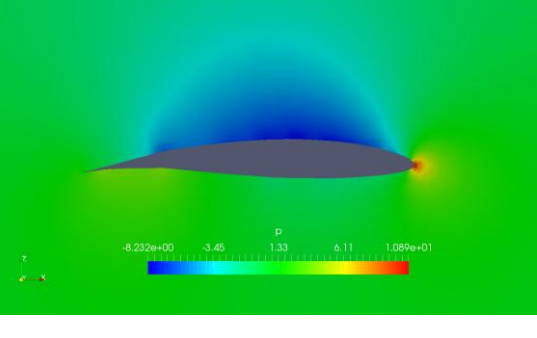
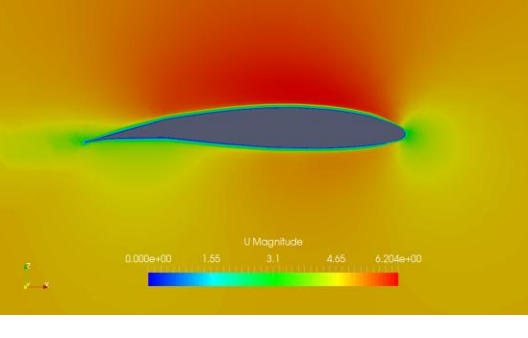
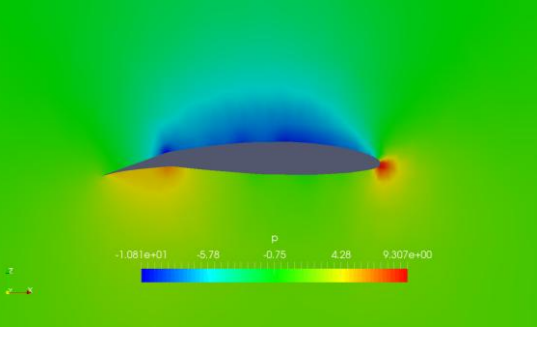
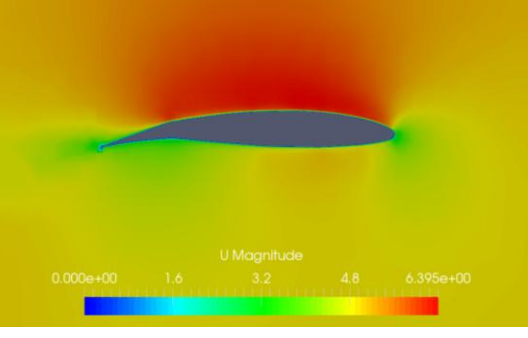
Figure 5.8: red plane represent the cutting plane for the foil section study



All results are reported in table 5.3.



Table 5.3: pressure and velocity results around a foil section

Flap's angle	Pressure Distribution	Velocity Distribution
<b>-10°</b>		
<b>-5°</b>		
<b>0°</b>		
<b>5°</b>		
<b>10°</b>		

#### 5.4.1 Comment on table 5.3

To explain results reported in table 5.3 it's necessary to divide the discussion for each geometry:

##### 0 Degrees

This is the starting case, the case without flaps. What is represented in the picture above is simply the profile of NACA 63412. As it's possible to see we have already lift in this case, this is possible because this is not a symmetric profile. The lift is explained by the pressure distribution, indeed on the suction side we have a depression zone while on the pressure side we obtain an higher pressure. This is deductible also by the velocity distribution, indeed we have a different velocity between pressure side and suction side, in the first one we have an high velocity while in the second one we have a low velocity.

##### -5 Degrees

This is a particular case, indeed as it's possible to see we obtain a similar pressure distribution between the suction side and the pressure side. This is due to the approach at the symmetric geometry of the profile. Although this, also in this case we have lift, a very little value but we have a positive lift. About velocity we have the same considerations with a distribution very similar to a symmetric profile (fig. 5.10- 5.11 ).

##### 5 Degrees

In this case we obtain a higher surface with low pressure on the suction side than the case with  $0^\circ$ . Moreover the absolute value of this depression zone is higher than the precedent case.

It's possible to see also a starting pressure growth beside the flap. The velocity close to the flap's pressure side is low.

##### -10 Degrees

Here we have a negative lift. This is the only case in our study which present this particularity, indeed the low pressure zone is collocated on the pressure side of the profile. Negative lift is due to the negative trailing edge flap's angle, we suppose that the border angle between positive and negative lift could be posed around  $-7.5^\circ$  degrees.

##### 10 Degrees

This case is the case where we reach the highest lift coefficient. The low pressure zone on the suction side is the biggest one and consequently the pressure difference is upper than the others cases.

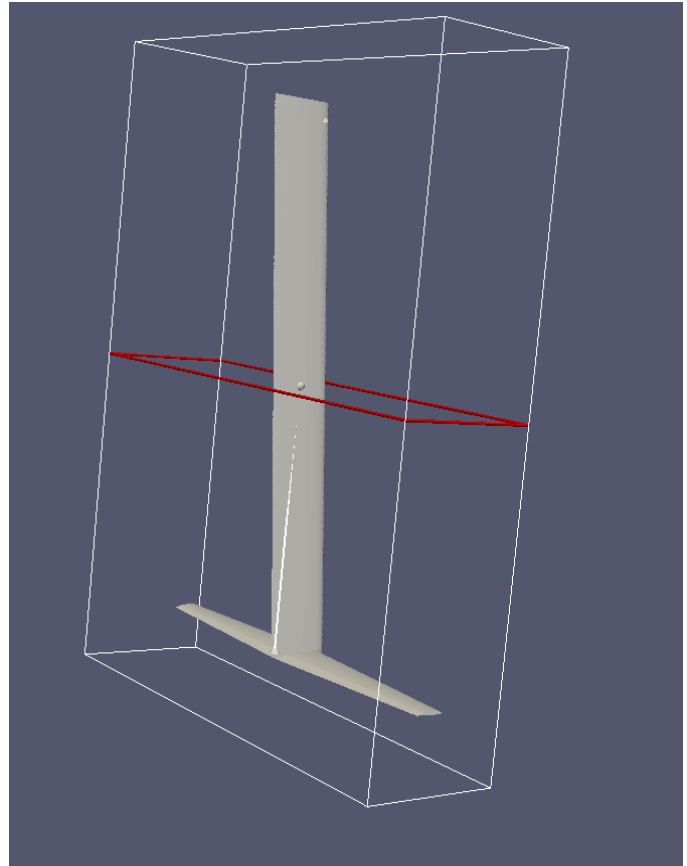
It's possible to see also an high pressure zone beside the flap which corresponds a low velocity zone.

#### 5.4.2 Flux around NACA 0012

To have a complete description of the phenomenon it's necessary observe also the pressure and velocity distribution around the symmetric profile of the centerboard.

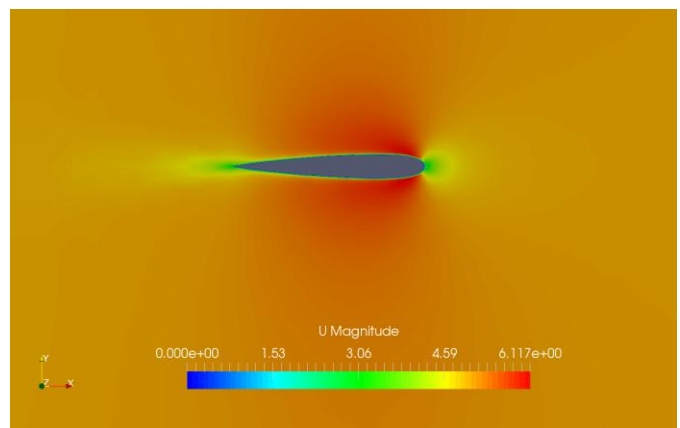
To obtain the correct slices, has been cut the geometry along the red plane represented in figure 5.9:

Figure 5.9: plane used to study NACA 0012



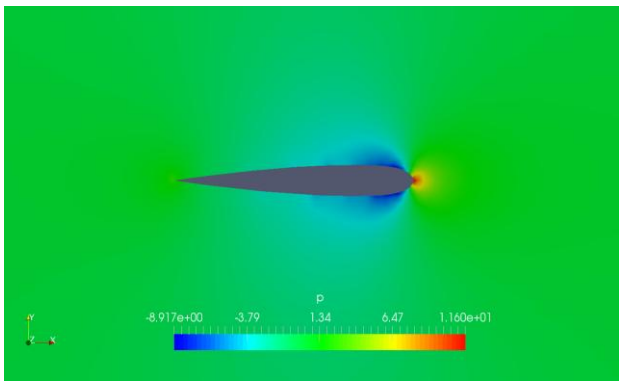
The results are reported only for one case ( $0^\circ$ ) and are showed in pictures below:

Figure 5.10: Velocity distribution along NACA 0012



As it's possible to see there is a perfect symmetric speed distribution around the profile, this of course causes also a symmetric pressure distribution as we can to see in figure 5.11

Figure 5.11: pressure distribution around NACA 0012



### 5.5 $C_L$ and $C_D$

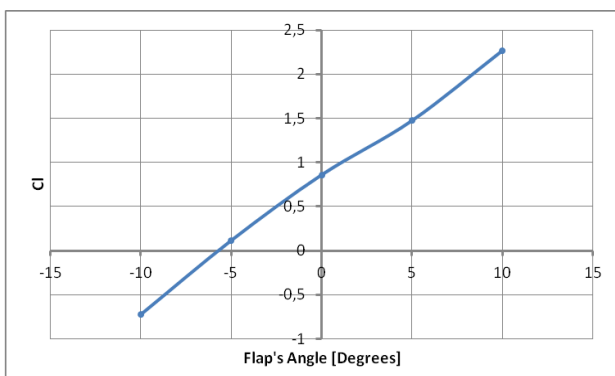
Drag and lift coefficients are reported for each case in the table below:

Table 5.4:  $C_L$  and  $C_D$

Flap's angle	$C_L$	$C_D$
-10°	-0,72737	0,263749
-5°	0,1138	0,2438
0°	0,859068	0,276287
5°	1,47561	0,319808
10°	2,270624	0,38357

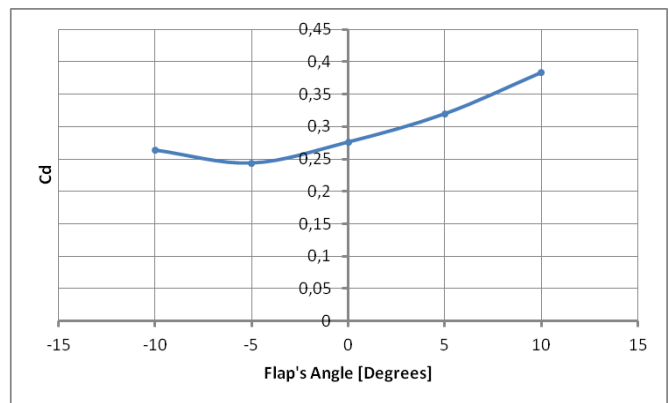
These are the results that we have obtained using parameters presented previously. To understand the phenomenon has been plotted some graphs:

Figure 5.12: lift coefficient dependence from flap's angle



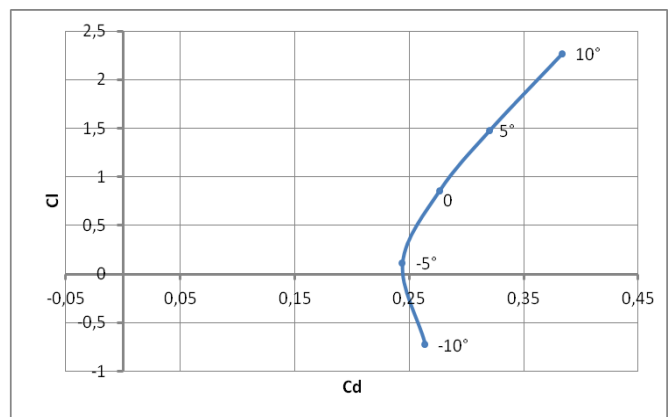
In the figure 5.12 is represented the lift coefficient trend. As showed with this kind of geometry and with this flux velocity, for a flap's angle larger than -5° we obtain a positive lift. For angles lower than -7° we have a negative lift.

Figure 5.13: drag coefficient dependence from flap's angle



Observing the graph above the minimum value of  $C_D$  is obtained for -5°.

Figure 5.14: drag and lift trend



The figure 5.4 it's useful to determinate the optimum flap's angle. Basing only on the graph the best angle is represented by -5°.

To find the minimum value needed of  $C_L$  for 5 m/s we supposed a total weight of 100 kg (boat + skipper). The result are founded by using equation 5.4.

Eq.5.4

$$C_L = \frac{2mg}{\rho c^2 A}$$

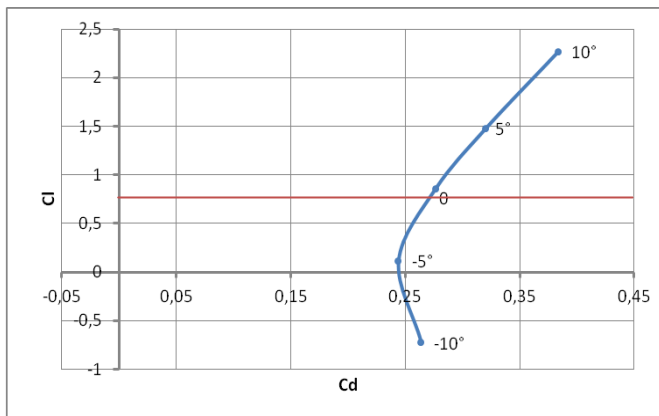
Where:

Variable	Mean	Unit	Value
$c$	Water Velocity	[m/s]	5
$A$	Approximated foil area	[m <sup>2</sup> ]	0,094
$\rho$	Water Density	[Kg/m <sup>3</sup> ]	1000
$mg$	Boat Weight Force	[N]	981

The solution give:

$$C_L = 0.834$$

Figure 5.15:



In the figure 5.15 the graph is divided in two regions by a red line. In the upper region  $C_L$  is sufficient to lift the hull and the skipper over the sea level, in the lower region no.

The required lift is achieved only for flaps angle higher than  $0^\circ$ . This is a reasonable value for a starting velocity of 5 m/s. We must remember that this kind of craft could achieve a velocity of 15/16 m/s so the value of  $C_L$  in this final case should be lower, around (with this geometry) 0.18.

## 6 Conclusion

The target of this work was to study the lift and drag generation in a centerboard of a foiling moth with a constant flux velocity of 5 m/s.

We obtained that :

- Lift is generated with a NACA 63412 in each case excepted the case with  $-10^\circ$  flap's angle.
- The angle with lower drag and a positive lift is  $-5^\circ$
- To lift a total weight of 100 kg ( boat + skipper) it is necessary work with a flap angle higher than  $0^\circ$  with a speed of 5 m/s

## 7 Future developments

This work used some simplifying hypothesis to make a first basic simulation of the problem.

Below is reported a list of some facts that can be considered in other works:

- The flap's angle is determined by the speed of the ship trough the sensor called wand (reported in figure 1.1. )
- The hull surging over the sea level causes a big drag fall which is not considered in this study.
- Improve the speed variation to study lift and drag in function of the hull's velocity.

## BIBLIOGRAPH

- [1] M. Colli, L. G. Lanza, R. Rasmussen, Assessing the accuracy of precipitation gauges: a CFD approach to model wind induced errors
- [2] A. May, The Foiling Guide
- [3] [www.cfd.direct.com](http://www.cfd.direct.com)
- [4] [www.cfd-online.co](http://www.cfd-online.co)
- [5] [www.openFOAM.com](http://www.openFOAM.com)

