Undergraduate Thesis

Title:
Computational analysis of sail designs for the efficient flow of autonomous sailboats

University of Crete,
Department of Physics

THEOFANIS GIOUMATZIDIS (ph4802)

Supervisors: GEORGIOS TSIRONIS AND GEORGIOS BARMPARIS

December-2022

Acknowledgments

I would like to express my gratitude to my main supervisor, professor of the Physics Department and Director of the Institute for Theoretical and Computational Physics of the University of Crete Mr. George P. Tsironis for the opportunity to work on an interesting project like this one and for his patience and guidance that have been invaluable.

I would like to express my gratitude to my second supervisor, Postdoctoral researcher of the Department of Physics of the University of Crete Mr. Georgios D. Barmparis for his input and feedback has been instrumental to this journey.

Lastly, I want to give many thanks, firstly, to my family and to my friends for the emotional support and courage that have given throughout this journey.

Abstract

The aerodynamics of a sail plays a crucial role on the speed and efficiency of a sailboat. For an autonomous sailboat the sail helps to have a fixed form in order to stable and predictable. By simulating a wind-tunnel on a CFD environment we have the ability to measure aerodynamics attributes like the Drag and lift coefficients of the sail with which we can evaluate them. In this thesis we have created a wind-tunnel environment, simulated using the CFD module of COMSOL Multiphysics. A good agreement between CFD and experimental or computational data is observed and the results are analyzed and discussed in details. A number of investigations have been conducted on the sail design demonstrating the best on the basis of Lift over Drag value for various angles of attack of each design iteration. The creation of a Wind-Water flow environment with a Sailboat sailing was followed with the environment not being stable enough to conduct sufficient measurements.

Contents

1 Introduction 6

2 Theory 7

2.1 Navier Stockes equations 7

2.2 Reynolds number 8

2.3 Drag and Lift Coefficient 9

3 Methodology 10

3.1 Introduction 10

3.2 Model instructions 10

3.3 Commentary 14

4 Validation and optimization of the wind tunnel 15

4.1 Sphere 15

4.2 Cylinder 19

4.3 Naca4415 Airfoil 21

5 Sail design investigation 25

5.1 Thickness investigation 25

Design 25

Results 26

5.2 Curvature of the Nose's Tip investigation 28

Design 28

Results 30

5.3 Nose Curvature investigation (y Direction) 32

Design 32

Results 32

5.4 Nose Curvature investigation (x Direction) 34

Design 34

Results 34

5.5 Camber investigation 37

Design 37

Results 38

5.6 Nose Curvature investigation (y Direction) with Cu = 2mm 39

Design 39

Results 39

5.7 Nose Curvature investigation (x Direction) with Cu = 2mm 41
Design 41
Results 41
5.8 Camber investigation with Cu = 2mm 43
Design 43
Results 44
5.9 Two-Part sail distance investigation 45
Design 45
Results 46
5.10 Two-Part sail between Angle investigation 47
Design 47
Results 48
5.11 Two-Part sail Camber investigation 49
Design 49
Results 49
5.12 Comparison of the one and two sail system designs 51
6 Optimal Design 52
6.1 Design Presentation 52
6.2 Model Instructions 52
6.3 Results 58
6.4 Use in real-time sailboat 59
7 Sailboat in Wind-Water flow environment 60
7.1 Introduction 60
7.2 Model Instructions 61
7.3 Commentary 72
8 Future Steps 73

1 Introduction

In order to create a stable and reliable autonomous sailboat that can navigate itself, a predictable and efficient sail is needed. A fixed form sail with simple geometry will eliminate any unpredictability a flexible one can bring because of its deformation. To make it efficient a physical investigation of different sail would be needed but would be difficult and very expensive so a CFD simulation was preferred.

Firstly, a wind tunnel environment was created using a block filled with air and a spatial sphere placed inside it. The air is given velocity and by the stress produced at the sphere's boundaries we extract the Drag Coefficient in regards to Reynolds number. The software variables for the wind tunnel environment were optimized and the model was verified by comparing the Drag Coefficient over Reynolds number curve with experimental and computational data. The model was verified also with a cylinder and with NACA4415 airfoil. The verification for the NACA4415 airfoil was conducted with Reynolds numbers 41000 and 82000 with the curves of Lift and Drag coefficients over the Angle of attack of the airfoil.

Secondly, a number of investigations optimizing the design were conducted by using the verified wind tunnel environment. The designs are drawn and tweaked at a 2D plane and then extruded to a 3D object. For each investigation we have a variable for every value of which we produce Lift and Drag coefficient data for a rage of Angle of Attack values. For each Variable value we also extract a max Lift over Drag value and compare them as it acts as the criterion for determining the better on. The best design is then presented as well as the instructions for creating the model that can produce its results.

Lastly, a Wind-Water flow environment with a Sailboat sailing was created. It is a model with time evolution of the flow of air and water interfering with a free moving material sailboat. The model makes the transition from stationary system to full speed flow but after a while it crashes before it can reach equilibrium. The model is presented as well as the instructions for its creation.

The software we are using is called COMSOL Multiphysics and is based on the Finite element method. This method is used to compute approximations of space- and time-dependent problems that are usually expressed in terms of partial differential equations (PDEs). These PDEs for the most cases cannot be solved analytically so the only way is to approximate a solution using numerical methods. The Finite element method has a wide range of applications in various branches of engineering such as Mechanical Engineering, Thermal and Fluid flows, Electromagnetic field, Biomathematics, Geo Mechanics, Business Management, etc. Our analysis is a fluid flow application that uses the Computational Fluid Dynamics (CFD) module that comes with COMSOL.

2 Theory

2.1 Navier Stokes equations

The Navier Stokes equations govern the motion of fluids and can be seen as the Newton's second law of motion for fluids. The Navier Stokes equations are defined as [1]:

$$\rho\left(\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} * \nabla \boldsymbol{u}\right) = -\nabla p + \nabla * \left(\mu(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T) - \frac{2}{3}\mu(\nabla * \boldsymbol{u})\boldsymbol{I}\right) + \boldsymbol{F}$$

where u is the vector for the fluid velocity
p is the fluid pressure
ρ is the fluid density
μ is the fluid dynamic viscosity

These equations We can break down the equation in four parts:

$\rho\left(\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} * \nabla \boldsymbol{u}\right)$	The inertial forces.
$-\nabla p$	The Pressure forces.
$\nabla * \left(\mu(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T) - \frac{2}{3}\mu(\nabla * \boldsymbol{u})\boldsymbol{I}\right)$	The viscous forces.
$\boldsymbol{F}$	The external forces applied to the fluid. [1]

Within the above equation, we can see four possible normalization scales that can be used to define a dimensionless Navier-Stokes equation. These are used to normalize the space, time, pressure, or flow variable; they also modify the gradient operator such that it defines a rate of change with respect to a dimensionless normalized variable.

Variable	**Normalized Variable**	**Normalized Gradient Operator**
Spatial coordinate r	r* = r/L	∇* = L∇
Flow rate u	u* = u/U	N/A
Time coordinate t	t* = t/(L/U)	N/A
Pressure p	High viscosity: p* = (pL)/(μU) High velocity: p* = p/(ρU2)	N/A

The normalization constant for the spatial variable and the time variable is arbitrary. For the spatial variable, the normalization constant could be some characteristic length scale in the system, such as the distance between two boundaries or the size of some particular feature in the system. A similar consideration can be made for the flow rate, such as the free-stream flow rate.

For the time variable, we could also apply a totally arbitrary normalization constant instead of using the flow rate and length scale as the basis for normalization. This would then enforce some requirement on the flow rate for a given length scale or vice versa. The resulting solution to the dimensionless Navier-Stokes equation will be invariant under any additional scale transformation in time because the Navier-Stokes equation is linear in time (assuming p is static or also linear in time).
Appling the linear transformations to the Navier-Stokes equation of motion we have the following non-dimensional result:

$$\frac{\partial \boldsymbol{u}^*}{\partial t^*} + (\boldsymbol{u}^*\nabla^*)\boldsymbol{u}^* = -\nabla^*\mathrm{p}^* + \frac{1}{Re}\nabla^{*2}\boldsymbol{u}^* + \frac{1}{Fr^2}\hat{g}$$

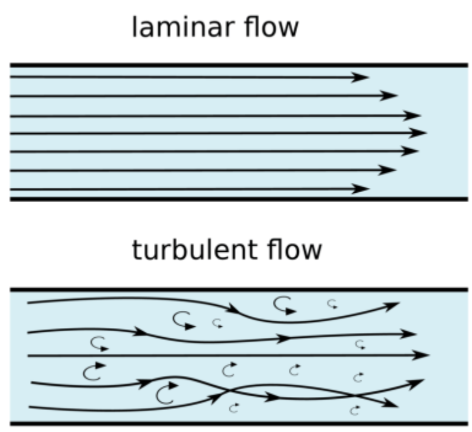
The dimensionless Navier-Stocks equations are defined by two dimensionless quantities: the Reynolds number (Re) and the Froude number (Fr) [2]. By ignoring gravity, we can ignore the last term and now the equations are defined only by the Reynolds number.

2.2 Reynolds number

The Reynolds number is the ratio of inertial forces to viscous forces within a fluid that is subjected to relative internal movement due to different fluid velocities. A region where these forces change behavior is known as a boundary layer, such as the bounding surface in the interior of a pipe. A similar effect is created by the introduction of a stream of high-velocity fluid into a low-velocity fluid, such as the hot gases emitted from a flame in air. This relative movement generates fluid friction, which is a factor in developing turbulent flow. Counteracting this effect is the viscosity of the fluid, which tends to inhibit turbulence [3]. The Reynolds number quantifies the relative importance of these two types of forces for given flow conditions, and is a guide to when turbulent flow will occur in a particular situation [4].
This ability to predict the onset of turbulent flow is an important design tool for equipment such as piping systems or aircraft wings, but the Reynolds number is also used in scaling of fluid dynamics problems and is used to determine dynamic similitude between two different cases of fluid flow, such as between a model aircraft, and its full-size version. Such scaling is not linear and the application of Reynolds numbers to both situations allows scaling factors to be developed.

With respect to laminar and turbulent flow regimes:

- laminar flow occurs at low Reynolds numbers, where viscous forces are dominant, and is characterized by smooth, constant fluid motion.
- turbulent flow occurs at high Reynolds numbers and is dominated by inertial forces, which tend to produce chaotic eddies, vortices and other flow instabilities [5].



A flow remains laminar as long as the Reynolds number is below a certain critical value. At higher Reynolds numbers, disturbances have a tendency to grow and cause transition to turbulence. This critical Reynolds number depends on the model, but a classical example is pipe flow, where the critical Reynolds number is known to be approximately 2000 [6].

The Reynolds number is defined as [7]:

$Re = \frac{r*V*L}{mu}$

Where r : The mass density of the fluid.

V : The flow speed of the object relative to the fluid

L : Characteristic length

mu : Viscosity coefficient

By keeping the density of the fluid, the characteristic length and the viscosity of the fluid fixed it becomes directly related to the flow speed.

2.3 Drag and Lift Coefficient

In fluid dynamics, the drag coefficient is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, such as air or water. It is used in the drag equation in which a lower drag coefficient indicates the object will have less aerodynamic or hydrodynamic drag. The drag coefficient is always associated with a particular surface area [8].

The drag coefficient C_D is defined as [9]:

$C_D = \frac{2*F_D}{\rho*A*V^2}$

F_d : The drag force, which is by definition the force component in the direction of the flow velocity.[9]

ρ : The mass density of the fluid.

V : The flow speed of the object relative to the fluid

A : Reference area.

Similarly, the lift coefficient (C_L) is a dimensionless quantity that relates the lift generated by a lifting body to the fluid density around the body, the fluid velocity and an associated reference area. A lifting body is a foil or a complete foil-bearing body such as a fixed-wing aircraft. C_L is a function of the angle of the body to the flow, its Reynolds number and its Mach number [10] [11].

The drag coefficient C_L is defined as [12]:

$C_L = \frac{2*F_L}{\rho*A*V^2}$

F_L : The Lift force

ρ : The mass density of the fluid.

V : The flow speed of the object relative to the fluid

A : Reference area

The Lift over Drag (L/D) Ratio

Because lift and drag are both aerodynamic forces, the ratio of lift to drag is an indication of the aerodynamic efficiency of the object. An object has a high L/D ratio if it produces a large amount of lift or a small amount of drag [13].

3 Methodology

3.1 Introduction

The purpose of this chapter is to present the digital wind tunnel environment. We are using the Comsol Multiphysics software with the CFD (Computational Fluid Dynamics) module. The model uses a 3D space and a stationary study and provides the Drag Coefficient at any Reynolds number. In order to model the fluid, we are using the **Laminar Flow** physics interface.
An interpretation of the environment is shown at the Figure 1. 1.

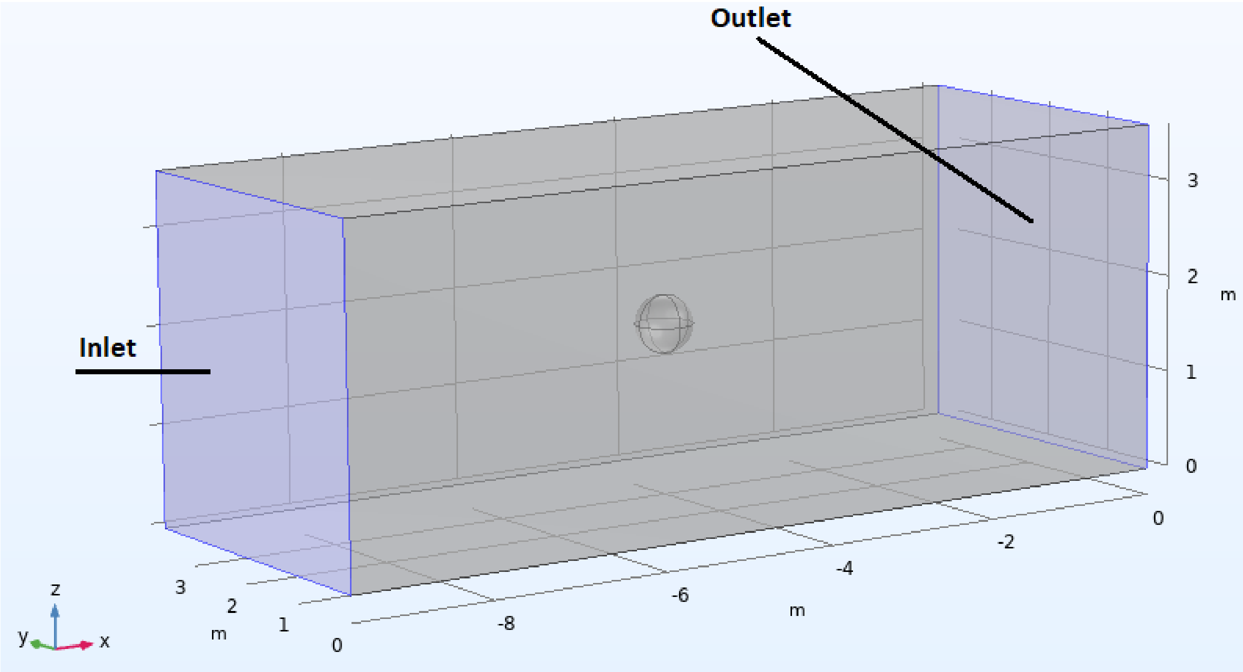


Figure 1. 1

3.2 Model instructions

The following Instructions are for the creation of a model that will produce the Drag Coefficient of a sphere at different Reynold number values.

New

In the **New** window, click **Model Wizard**.

Model Wizard

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow> Single-Phase Flow>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

Global Definitions

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
rho_Air	1.225[kg/m^3]	1.225 kg/m^3	Air Desity
mu_Air	1.802E-5[Pa*s]	1.802E-5 Pa*s	Air viscosity coefficient
L	0.6[m]	0.6 m	Characteristic Length
Re	10000	10000	Reynolds number
V_Air	Re*mu_Air/(rho_Air*L)	0.24517 m/s	Airflow speed

Default Model Inputs

1 In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.
2 Locate the **Requested Model Inputs section**.
3 At the **Temperature** row, under the **Expression** column, wright 288.15[K].

Geometry 1

1 At the **Graphics** window click at the **Transparency** button.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type 4*8*L.
4 In the **Depth** text field, type 4*3*L.
5 In the **Height** text field, type 4*3*L.

Sphere 1 (sph1)

1 In the **Geometry** toolbar, click **Sphere**.
2 Locate the **Size** section. At the **Radius** test field type L/2.
3 Locate the **Position** section. Fill the test fields as:

11*L	**x**
6*L	**y**
6*L	**z**

Difference 1 (dif1)

2 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
3 Locate the **Objects to add** section and click **Paste Selection**.
4 In the **Paste Selection** dialog box, type blk1.
5 Click **OK**.
1 Locate the **Objects to subtract** section. Click **Paste Selection**.
6 In the **Paste Selection** dialog box, type sph1.
7 Click **OK**.
8 At the top of the **Settings** window click **Build All Objects**.

Laminar Flow (spf)

Inlet Air

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.
2. At the **Label** text field type Inlet Air.
3. Locate the **Boundary Selection** section and click **Paste Selection**.
4. In the **Paste Selection** dialog box, type 1.
5. Click **OK**.
6. Locate the **Velocity** section and at the text field for $\mathbf{U_0}$ type V_Air.

Outlet

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Outlet**.
2. Locate the **Boundary Selection** section and click **Paste Selection**.
3. In the **Paste Selection** dialog box, type 14.

Symmetry 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Symmetry**.
2. Locate the **Boundary Selection** section and click **Paste Selection**.
3. In the **Paste Selection** dialog box, type 2-5.
4. Click **OK**.

Materials

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in>Air**.
4. Click **Add to Component** in the window toolbar.
5. Close the **Add Material** window.

Air (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.
2. Locate the **Material Contents** section.
3. At the **Dynamic viscosity** row, under the **Value** type wright mu_Air.
4. At the **Density** row, under the **Value** column type rho_Air.

Mesh 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1.**
2. Locate the **Physics-Controlled Mesh** section. From the **Element size** list, choose **Fine**.

Study 1

Parametric Sweep

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Study 1** and choose **Parametric Sweep**.
2. In the **Settings** window for **Parametric sweep**, locate the **Study Setting** section.
3. Click **Add**.
4. Under the **Parameter** column, from the list choose **Re(Reynolds number)**.
5. Under the **Parameter value list**, type 1, 10, 100, 1000, 10000, 100000, 1000000.

Job Configurations

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Study 1** and choose **Show More Options….**
2. Select the **Study>Solver and Job Configurations** check box.
3. Click **OK**.
4. In the **Model Builder** window, under **Component 1 (comp1)>Study 1** right-click **Job Configurations** and choose **Parametric Sweep**.
5. Locate the **General** section. From the **Defined by study step** list, choose **Parametric Sweep.**
6. In the **Model Builder** window, under **Component 1 (comp1)>Study 1>Job Configuration** right-click **Parametric Sweep 1** and choose **Enable**.
7. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Study 1** and choose **Get Initial Value**.
8. In the **Model Builder** window, under **Component 1 (comp1)>Study 1>Job Configuration** click **Parametric Sweep 2.**
9. Locate the **Error** section. Unselect the **Stop if error** check box.

Results

Cross Section

1. In the **Model Builder** window, under **Component 1 (comp1)>Results** right-click **Derived Values** and at the **Integration** option, choose **Surface Integration**.
2. At the **Label** text field type **Cross Section**.
3. Locate the **Selection** section. Click **Paste Selection**.
4. At the **Paste Selection** text dialog box, type 6-13.
5. Click **OK**.
6. Locate the **Expressions** section. Under the **Expression** column type -nx*(nx<0).
7. At the top of the **Settings** window, click **Evaluate**.

Drag Coefficient

1. In the **Model Builder** window, under **Component 1 (comp1)>Results>Derived Values** right-click **Cross Section** and choose **Duplicate**.
2. At the **Label** text field type Drag Coefficient.
3. Locate the **Expressions** section. Under the **Expression** column replace the -nx*(nx<0) expression with -spf.T_stressx*2/(rho_Air*V_Air^2* (0.27658[m^2])).

Compute

1. In the **Model Builder** window, under **Component 1 (comp1)>Study 1** click **Parametric Sweep**.
2. Click **Compute**.

Plot results

1. In the **Model Builder** window, under **Component 1 (comp1)>Results>Derived Values** click **Drag Coefficient**.
2. Locate the **Data** section. At the **Dataset** list, choose **Study 1/Parametric solution 1**.
3. At the top of the **Settings** window, click the arrow next to the **Evaluate** button and choose **New Table**.
4. In the down-right window, at the **Table 2** tab click **Table Graph**.
5. In the **Graphics** window click at the **x-Axis Log Scale** and **y-Axis Log Scale** buttons.

For a better view of the results, Origin Lab or Excel are recommended.

A computed version of this model can be found on:
dropbox.com/scl/fo/361lf7jl80prwqn42l997/h?dl=0&rlkey=abx80noagehwnzt5nw7mhnm7n

3.3 Commentary

After the Difference operation we end up with one domain that consist of the block minus the sphere that will act as our fluid domain.
The **Laminar Flow (spf)** interface is used to compute the velocity and pressure for the flow of a single-Phase fluid in the laminar flow regime. With this interface we will create the flow by introducing an inlet and an outlet and assign velocity to the fluid. We take advantage of the high degree of symmetry of the sphere by using Symmetry Wall condition at the remaining block boundaries so that the model can get the result with a lot less computations. At the other designs that don't have high degree of symmetry we will leave the default Wall no slip condition. The sphere (or any object we test at the wind tunnel) we leave the default Wall no slip condition. We use incompressible flow at the x-Axis direction and the outlet provides no resistance.

We use physics-controlled mesh as the geometry of the problem is simple enough and the automating meshing does a sufficient job.
After the computation, at the Derived Values subsection of the Results section, with a surface integration of the sphere boundaries we extract the drag Coefficient, integrating the formula:

$$\frac{-\mathrm{spf.T\_stressx} * 2}{\mathrm{rho\_Air} * V^2 * A}$$

Where: $\mathrm{spf.T\_stressx}$ is the pressure force at the x direction
$\mathrm{rho\_Air}$ is the air density
V the air speed
$A = \frac{pi*L^2}{4}$ is the cross-section of the sphere
L the sphere diameter

For the design investigations, in order to create the design, we sketch the design in 2D and then we extrude the design to 3D. We use a Work plane to create a 2D space that we can sketch and to extrude the design we use a tool called Extrude.
All models were built in COMSOL Multiphysics 6.0.

4 Validation and optimization of the wind tunnel

At this chapter we want to validate the procedure using existing experimental and computational data by using as stable and fast computation. The aim is to confirm the behavior of Drag coefficient not a comprehensive analysis as the application we will use will be comparative.

4.1 Sphere

Using experimental Curve of Drug Coefficient for the sphere given below we will investigate certain attributes of the model.

$$C_D = \frac{24}{Re} + \frac{2.6\left(\frac{Re}{5.0}\right)}{1+\left(\frac{Re}{5.0}\right)^{1.52}} + \frac{0.411\left(\frac{Re}{2.63*10^5}\right)^{-7.94}}{1+\left(\frac{Re}{2.63*10^5}\right)^{-8.00}} + \frac{0.25\left(\frac{Re}{10^6}\right)}{1+\left(\frac{Re}{10^6}\right)}, [14]$$

This formula has been accurate up to $Re = 10^6$

We plot the formula below at the Figure 4.1. 1

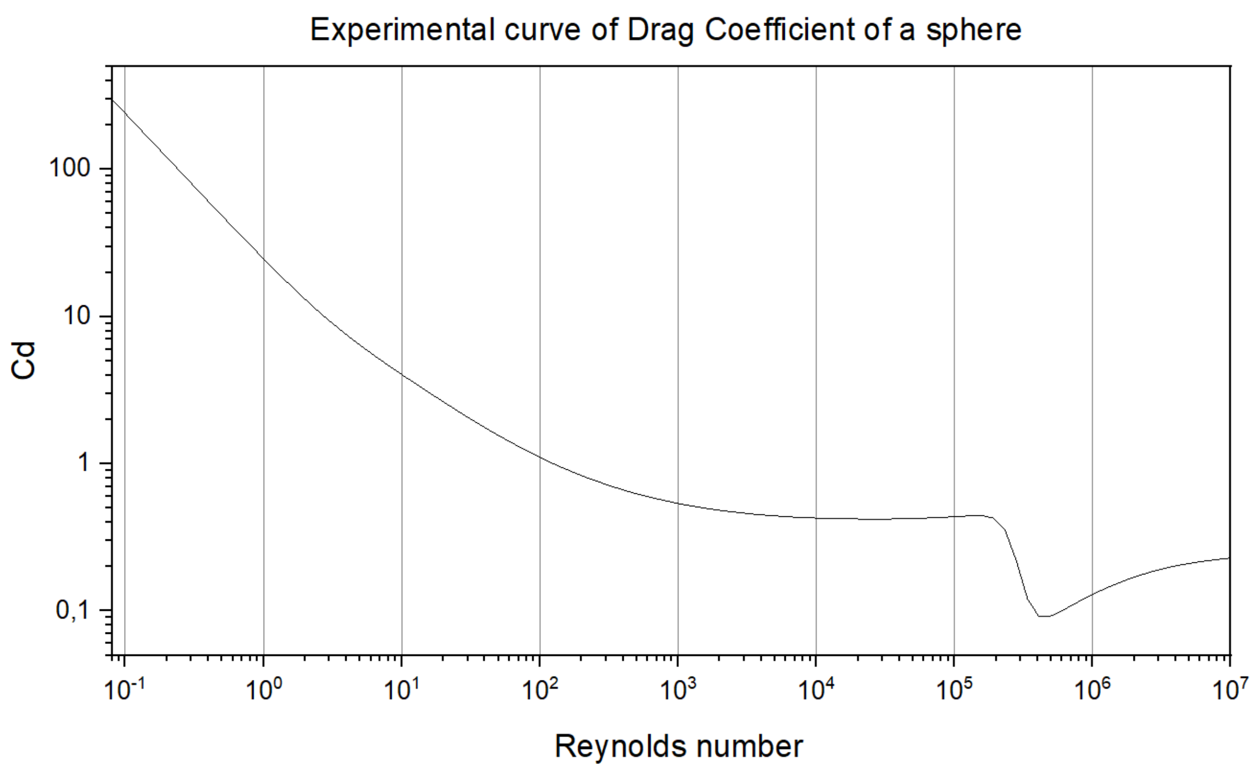


Figure 4.1. 1

First implementation

With a first implementation of the model described at the methodology chapter using Normal mesh refinement, a sphere radius of r=0.3 m, block dimensions: 1.8 m x 1.8 m x 4.8 m and the sphere at 1.8 m distance from the inlet we have the results portrayed at Figure 4.1. 2 below

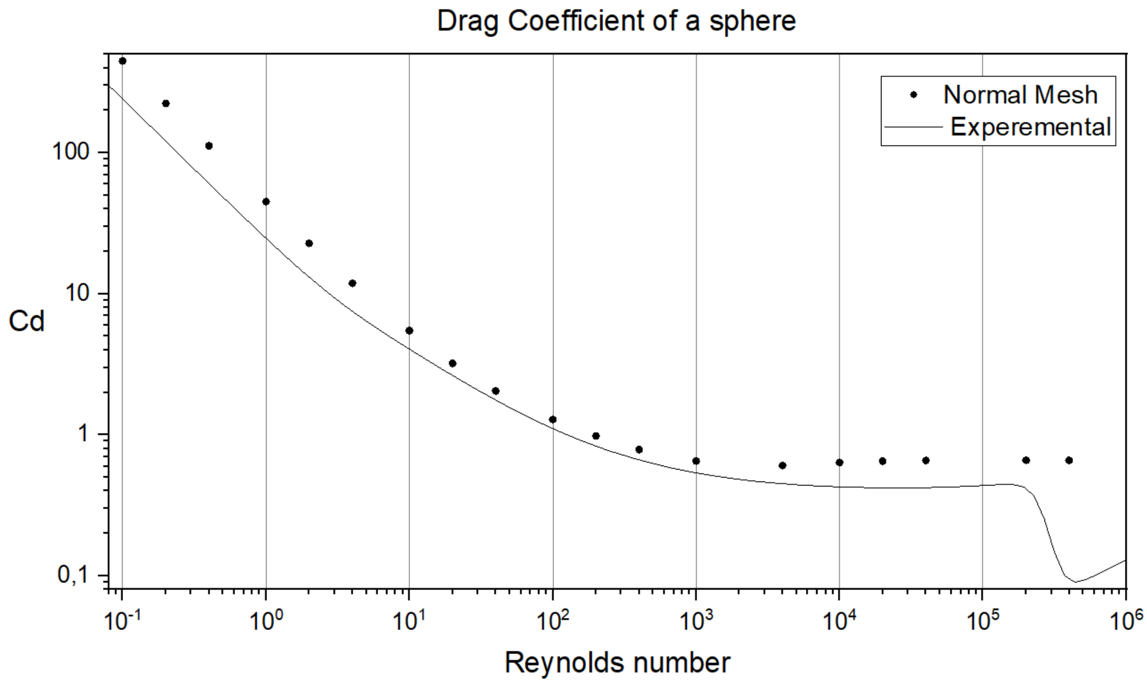


Figure 4.1. 2

It's clear that the curve is off and probably because the Normal mesh is too sparce.

Mesh refinement investigation

Running the same values for fine and finer mesh refinement putted with the previous data we have the following results portrayed at Figure 4.1. 3.

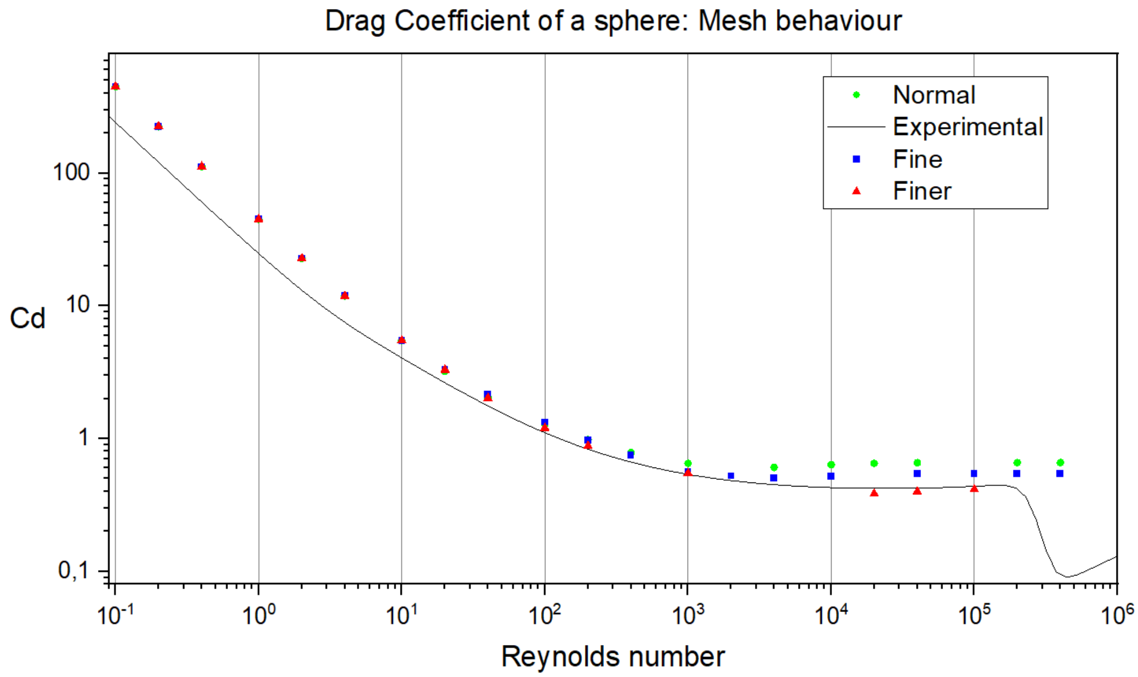


Figure 4.1. 3

There is improvement in the biggest numbers of Re. This is expected because as the Re is being increased, increases also the speed that needs finer detail.
The error at low values of Re indicates that probably the Symmetry walls interfere measurably with the airflow but as the speed increases that factor becomes negligible. A solution can be to increase the size of the block or to use different kind of walls.
There is an increasing number of runs not converging as the mesh becomes finer. This can probably be solved with a bigger block.
The drop of the curve at around $6x10^5$ is not being followed by the results. This is because the flow with Reynolds number value over 3500 is defined as Turbulent. The models presented have pretty good results even using Laminar flow until around Re $=10^5$. We will keep using Laminar flow optimizing the model with data with upped limit Re $=2x10^5$ as with the turbulent flow physics interface we have convergence issues.

Wall interference investigation

Using Finer mesh refinement, we will run the model with bigger block as we can see at the Figure 4.1. 4. At the Figure 4.1. 5 the label “Finer” refers to the model with the same block dimensions as before (1.8m x 1.8m x 4.8m) and the multiplier refers to the multiplication of these block dimensions. We also adjust the distance of the sphere accordingly (d =1.8m for the Finer, d = 3m for the 2xFiner, d = 6.6m for the 4xFiner).

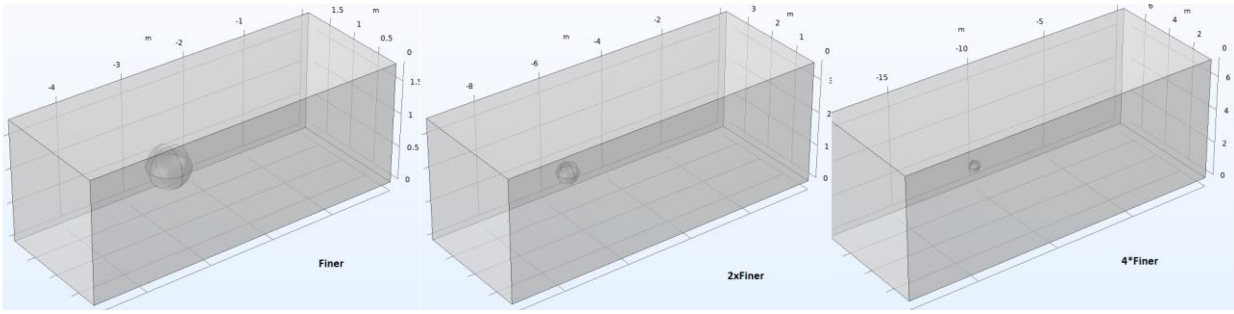


Figure 4.1. 4

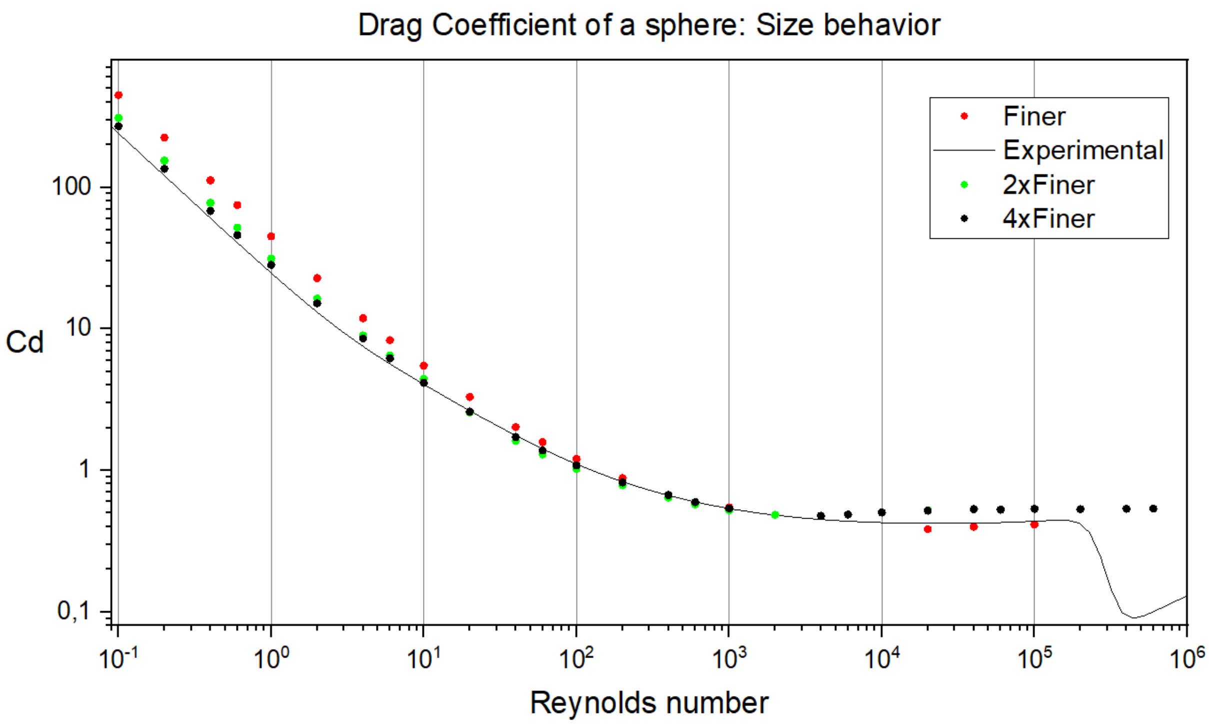


Figure 4.1. 5

As the dimensions of the block are increased, the results curve converges to the experimental curve but also with greater computational time expense. The relation reverses at the critical point at about Re = 3500 where the flow is becoming turbulent at internal flow situations. The Laminar flow interface gives the expected behavior evet at Reynolds numbers bigger that 3500 until around Re = $2 * 10^5$ which is the critical point at external flow situations. Using the turbulent flow physics interface we have convergence issues.

Given that, we will assume that we have an external flow situation and continue using Laminar flow. Also, the bigger the block the less runs fail to converge. This means that indeed the walls were interfering and we can solve the problem by increasing the block size.

4.2 Cylinder

Here we present Computational and experimental curves that we will use to compare our results [15]. Using the data from the Figure 4.2. 1 below with WebPlotDigitizer we extracted and compared them with ours.

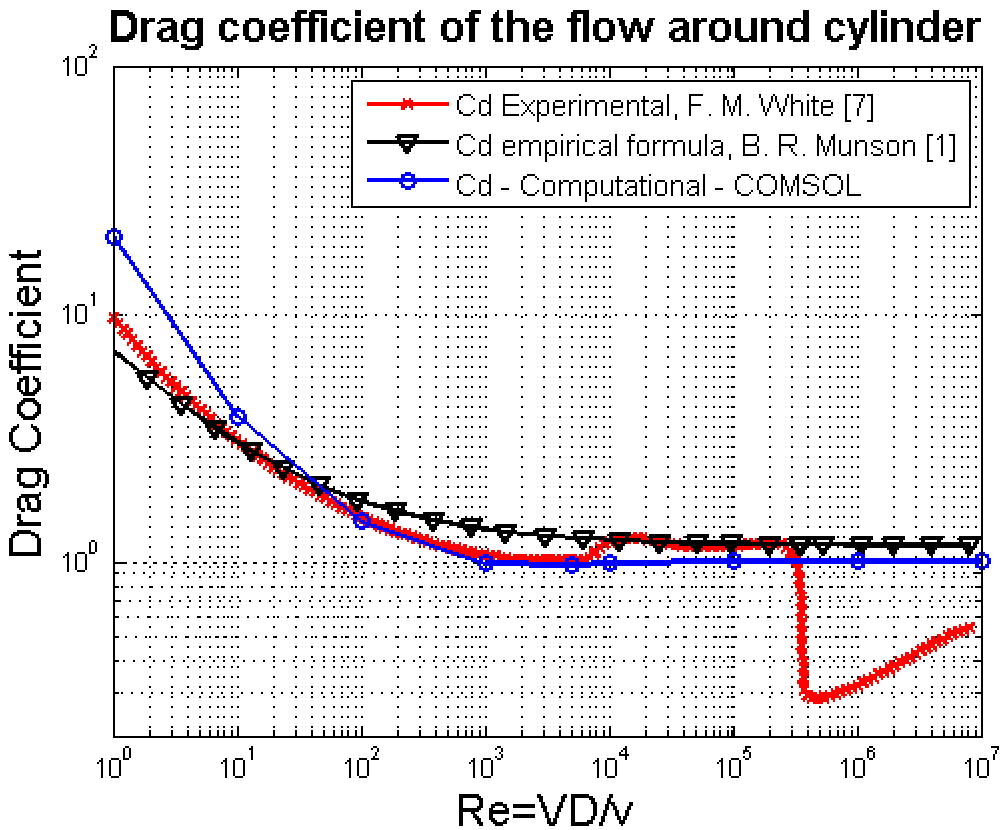


Figure 4.2. 1

Taking into account the results of the previous investigations with the sphere, we construct a model with a cylinder with a radius of r = 0.3m and height 1.2m and a block dimensions 28,8m x 28,8m x 57,6m (Figure 4.2. 2) and a Fine mesh refinement. We are still using Laminar flow and this time we will use as boundary conditions the Wall boundary condition with no slip wall condition. The results are portraited at Figure 4.2. 3

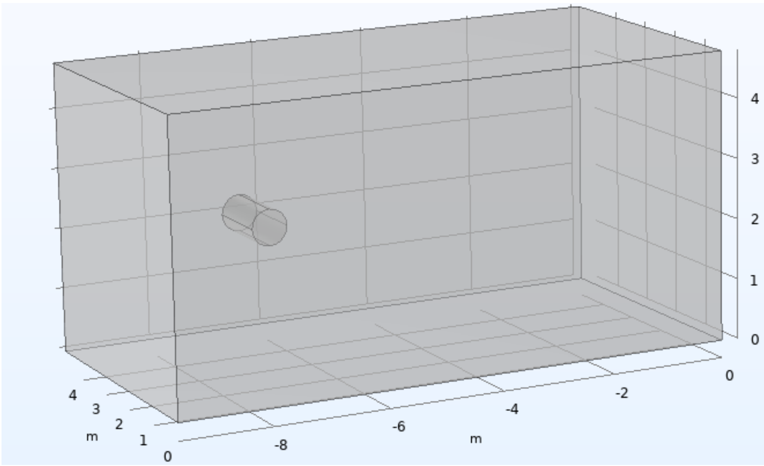


Figure 4.2. 2

*Not in scale

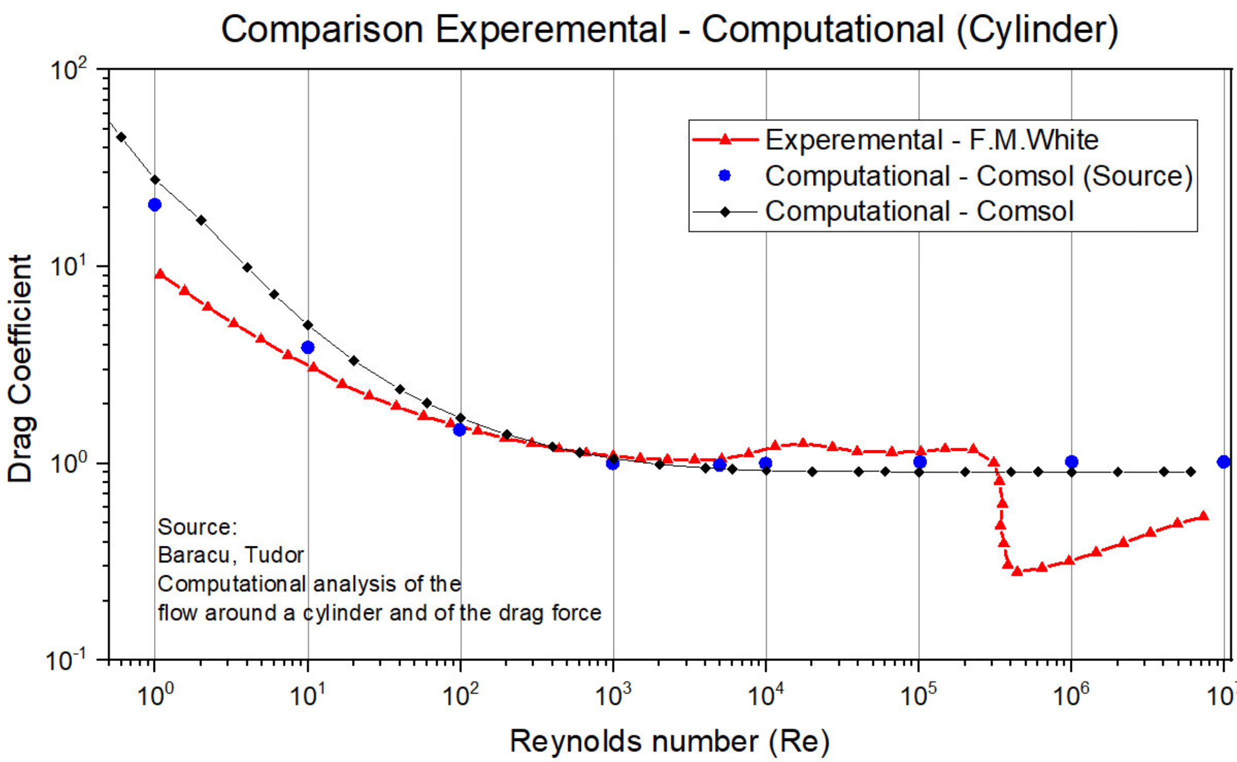


Figure 4.2. 3

Up until Re=100 the results are off but at the area between Re=100 and Re=$2 * 10^5$ the results are satisfactory. At Re=$2x10^5$ is the critical point of the External flow so again it seems like the assumption that the flow acts as external is correct.

4.3 Naca4415 Airfoil

The Naca4415 Airfoil design is a design with specific ratios of dimensions and is one of the designs that are being used as a reference for research and development applications. A 2D diagram is show at the Figure 4.3. 1 below:

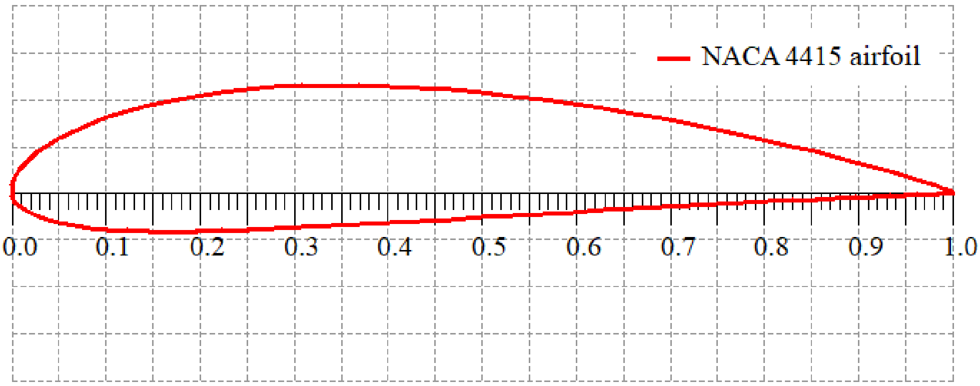


Figure 4.3. 1

We will use an experimental Curve of Lift and Drug Coefficient to compare with our results [16]. Using the data from the figure below with WebPlotDigitizer we extracted and compared them with ours. The following data at Figure 4.3. 2 are for fluid speed of 3m/s and Reynolds number value Re=41000.

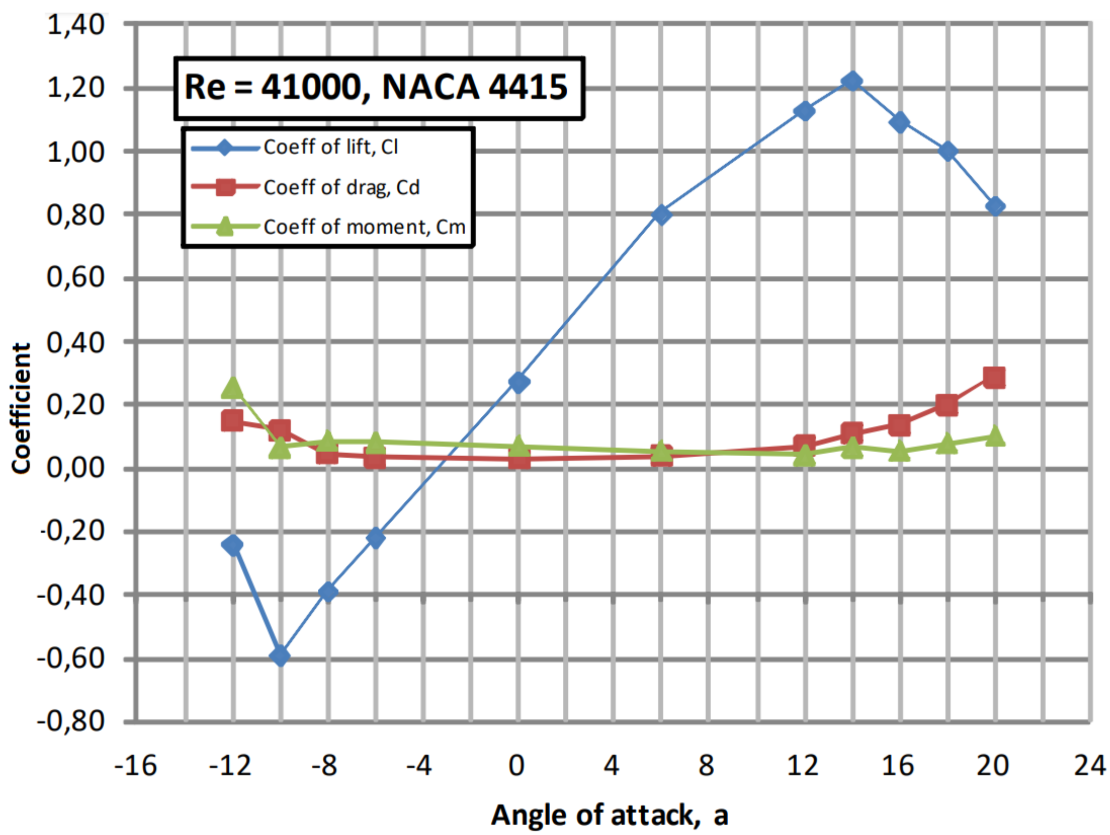


Figure 4.3. 2

The fluid that is being used here is not air so for the same Reynolds number (41000) with air as the flow fluid the fluid speed will be 1,2062 m/s.
Taking into account the results of the previous investigations we construct a model with a NACA4415 airfoil with length 1,716m and Chord length 0,85m. The block dimensions are (7,2m x 7,2m x 18m) and the airfoil is located 3m from the inlet. We use Normal mesh refinement and we will use as boundary conditions the Wall boundary condition with no slip wall condition. The results are portraited at Figure 4.3. 3

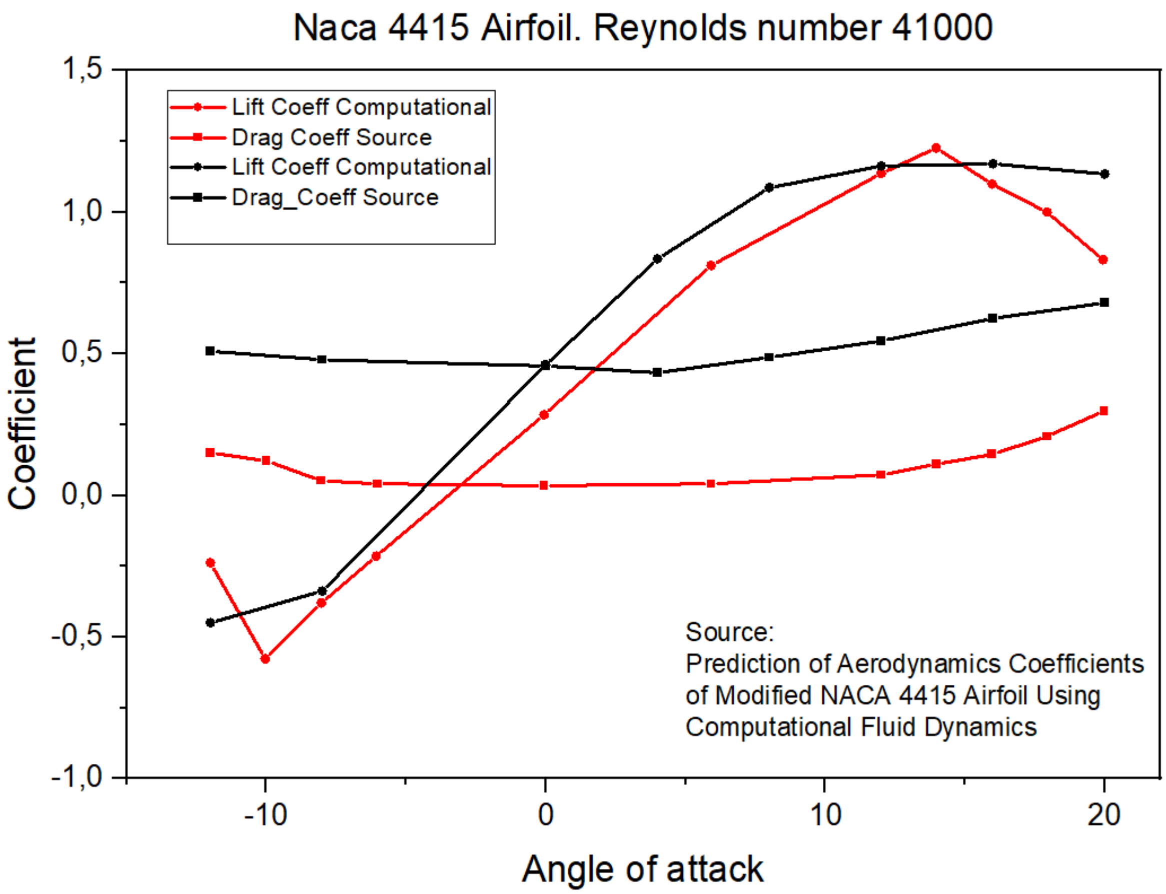


Figure 4.3. 3

The lift coefficient curve is pretty close to overlapping with that Computational curve of the referenced paper. The Drag Coefficient curve has the same behavior as the referenced paper's but is displaced vertically about 0,4.

The following data at Figure 4.3. 4 are for fluid speed 18m/s and Reynolds number value Re=82000.

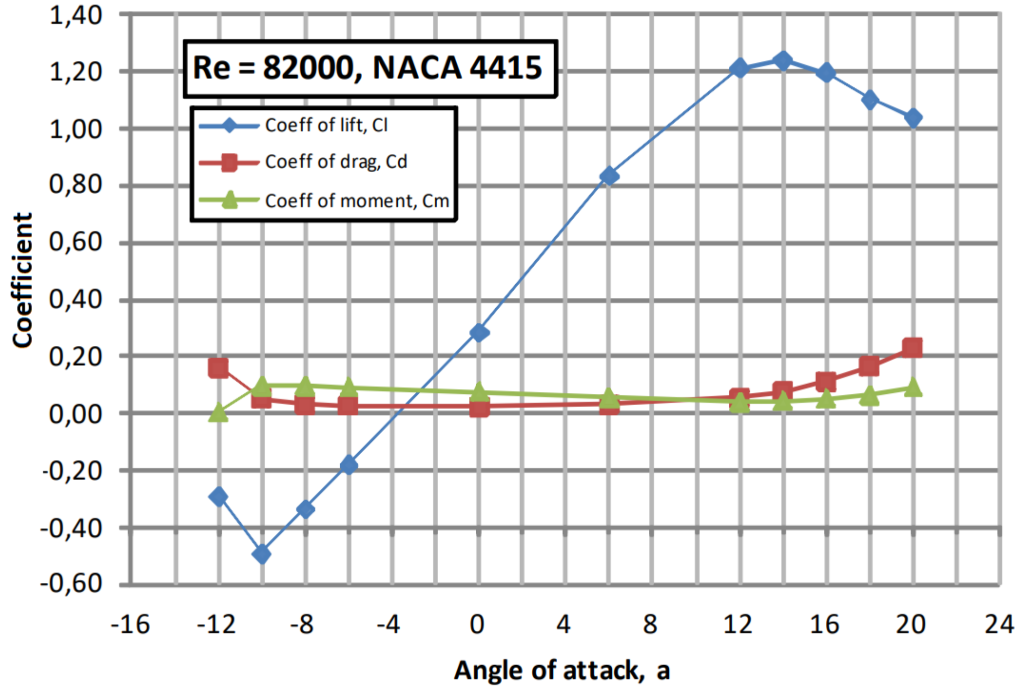


Figure 4.3. 4

For air as the flow fluid with Reynolds number 82000 the wind speed is 2,4125 m/s. Using the same setup as before we have the following results at Figure 4.3 5:

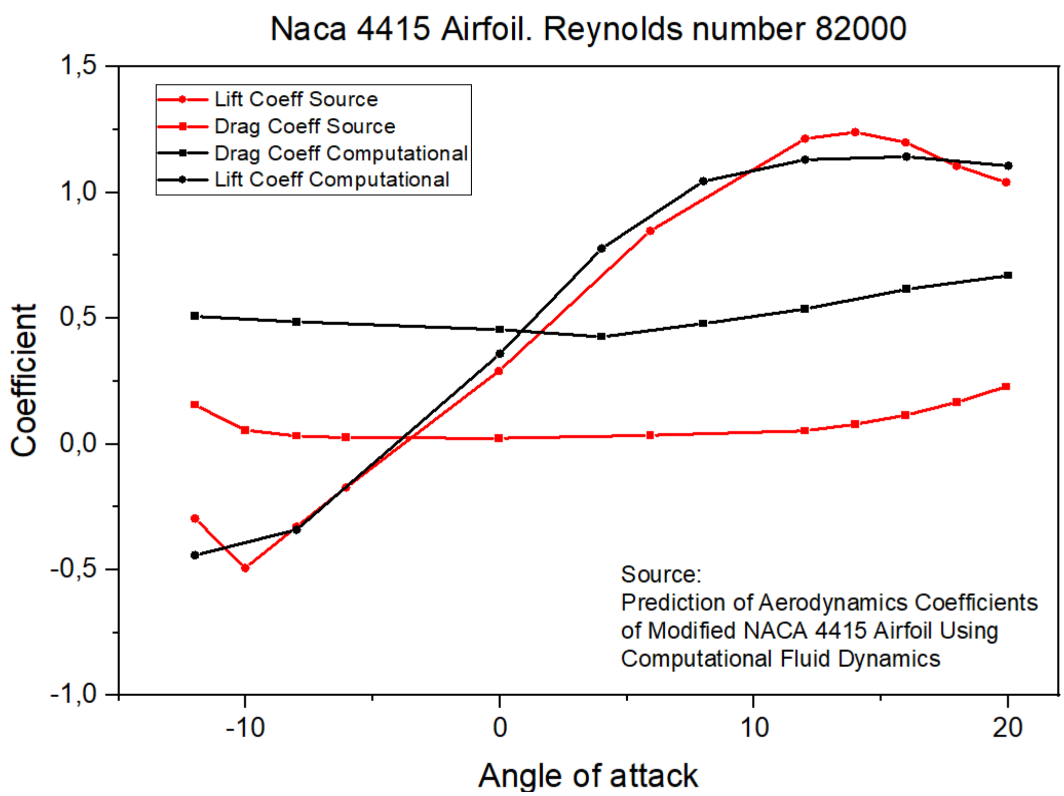


Figure 4.3. 5

We have the same behavior as before.

4.4 Conclusions

- The finer the mesh refinement we have more accurate results but with more non convergence runs.
- The Bigger the block size, the smaller the Boundary interference more accurate results.
- Keeping the same mesh refinement and adjusting the Block size we can see that the mesh elements directly at the object becomes smaller. The smaller the mesh elements directly at the object, the more the Turbulent feature become instrumental and the results get worse.
- The Laminar flow physics interface gives the expected behavior evet at Reynolds numbers bigger that 3500 until around Re = $2 * 10^5$ and the Turbulent flow physics interface has convergence issues so we will continue to use Laminar flow. The reasoning is that we will assume that the flow is external so the critical point the flow becomes Turbulent is around Re = $2 * 10^5$.
- The results with the Cylinder reinforce the assumption that the flow acts as external.
- Using the Naca4415 airfoil we have accurate results for Lift Coefficient for Re 41000 and 82000 and accurate behavior for Drag Coefficient with a displacement of the value vertically of 0,4.
- The Laminar flow interface with the setup used for Naca4415 seems to simulate satisfactorily the environment for the purposes of this analysis.

5 Sail design investigation

We want to identify how different design attributes interfere with the lift and drag of the sail.
For these investigations we will use a block with dimensions 4m x 4m x 10m and the sail will be 1,667m from the inlet. The block dimensions are proportional of the Chord length that is L=0,5m. The Reynolds number is 82000.

5.1 Thickness investigation

Design

The design consists of an eclipse representing the nose and two Quadratic Bezier curves that start from the eclipse and converge at the end of the airfoil representing the tail. The one radius equals to D/2 and the other to D keeping the thickness variable equal to D and certain scale characteristic as the D changes. A 2D representation is show below at the Figure 5.1. 1 and 2D representations with different thickness values are shown at Figure 5.1. 2.

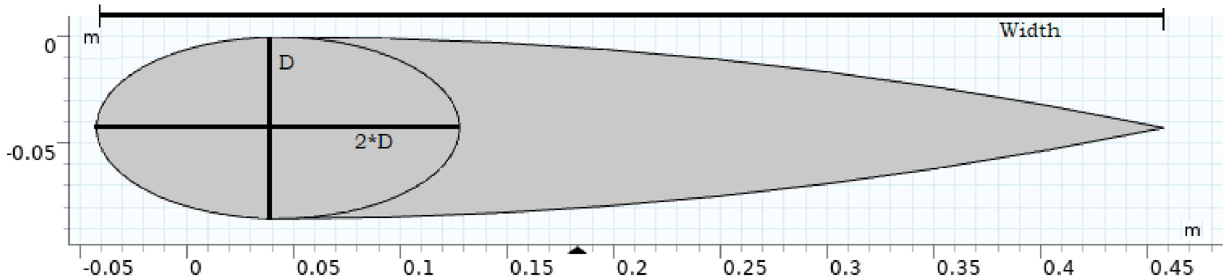


Figure 5.1. 1

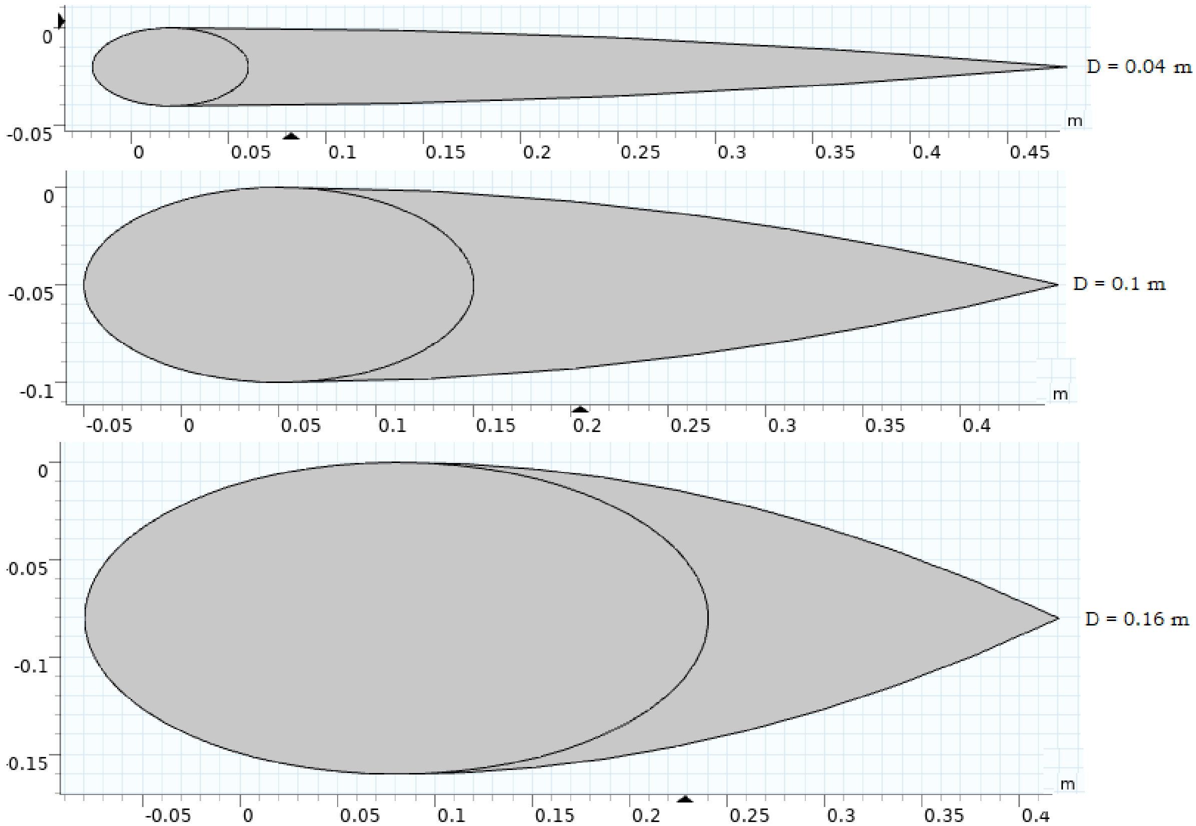


Figure 5.1. 2

The designs are extruded to 3D objects

<u>Dimensions</u>

Width (chord length) = 0.5 m

Airfoil height = 1.20 m

$D = (4 - 16)cm$ (varied)

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0^o to 35^o with 5^o step and as a metric for our results we use the Lift over Drag value that is the maximum for each variation of the variable D.

For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.1. 3:

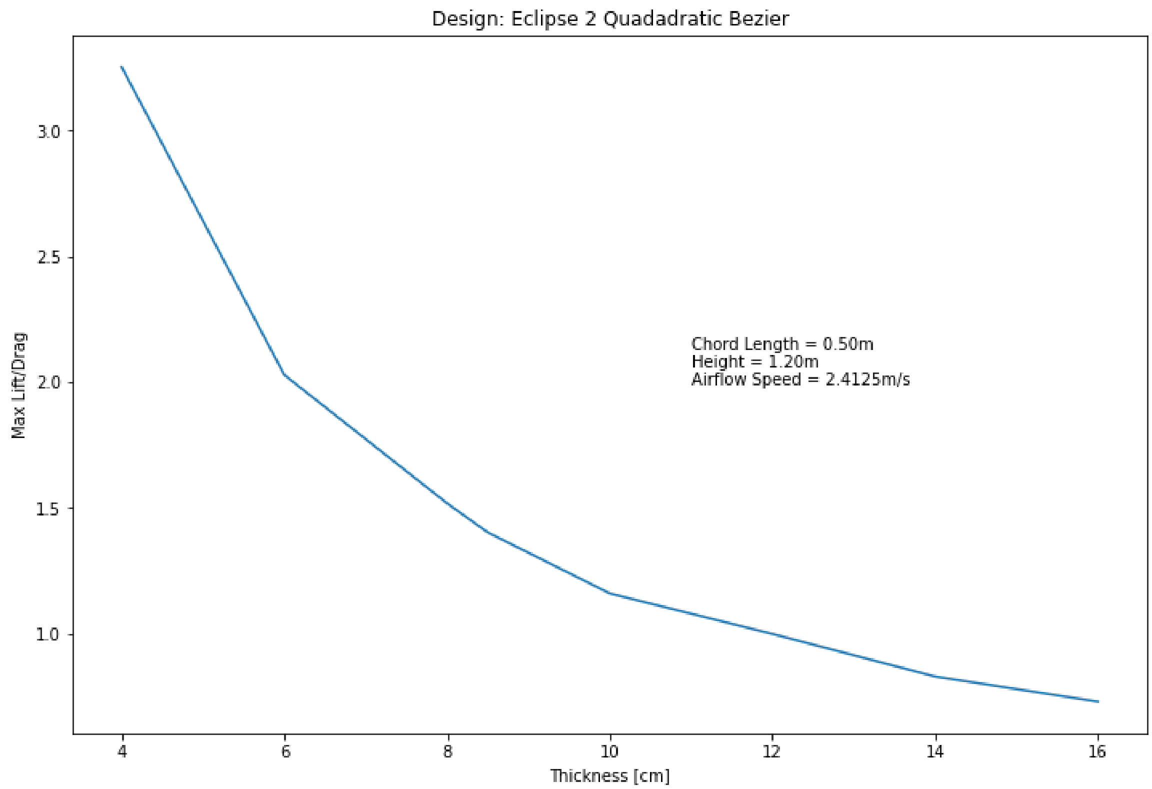


Figure 5.1. 3

At 12cm thickness and less there is a beneficial Lift to Drag ratio at least at the max value of the angle of attack. This critical point is probably higher as the Drag coefficient value is higher than it should be as it is shown at 2.3_Naca4415 Airfoil.

It is clear that the smaller the thickness the biggest the Lift over Drag value. For construction purposes we will use thickness of D=8.5cm.

For D=8.5cm we have the following curves for Lift and Drag Coefficients portrait at Figure 5.1. 4:

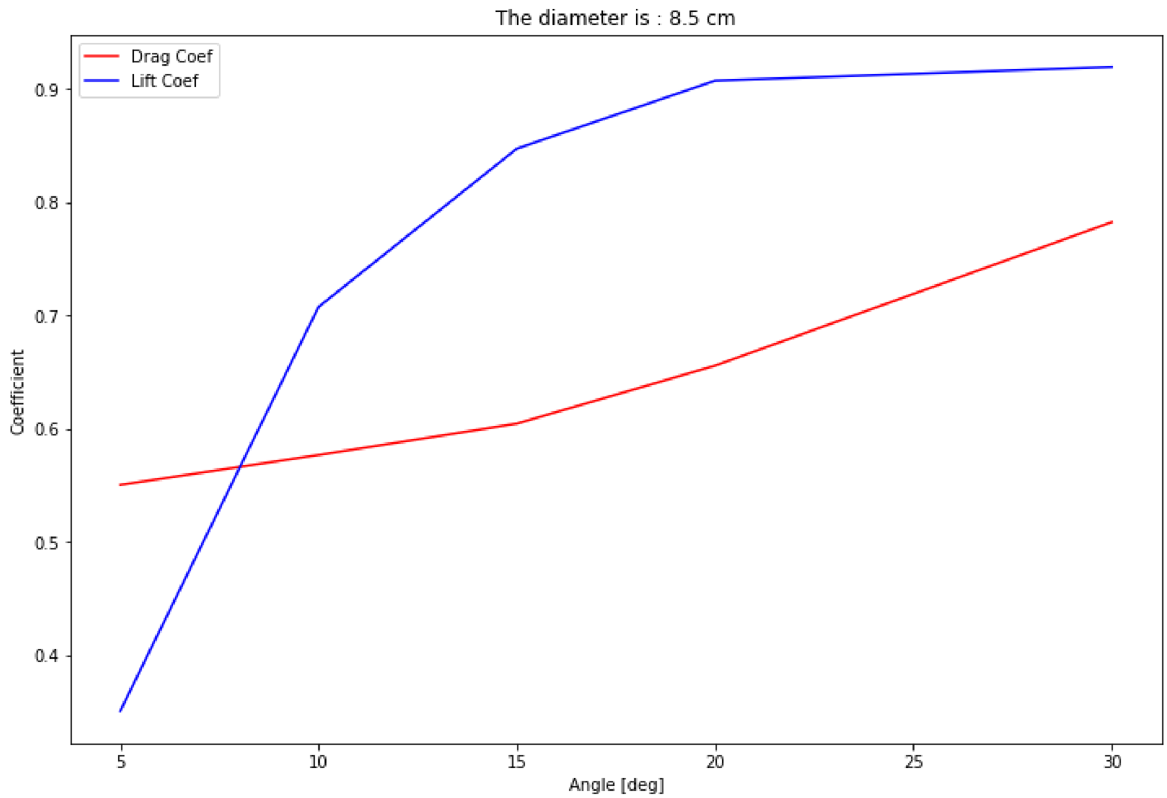


Figure 5.1. 4

The max Lift over drag value is 1,402

5.2 Curvature of the Nose's Tip investigation

Design

The design consists of four Quadratic Bezier curves. Two that are connected at the nose's tip and at the opposite sides of an imaginary circle located at the max thickness position with radius a2. The other two are the reflection of the first and they end and are connected at the tip of the tail. The connections at the imaginary circle are been smoothed by a tool called Fillet with radius 0.1 m and the connection at the tip of the nose as well but with the radius as a variable. A 2D representation of the initial and smoothened design is show below at the Figure 5.2. 1 and 2D representations with different curvature values are shown at Figure 5.2. 2.

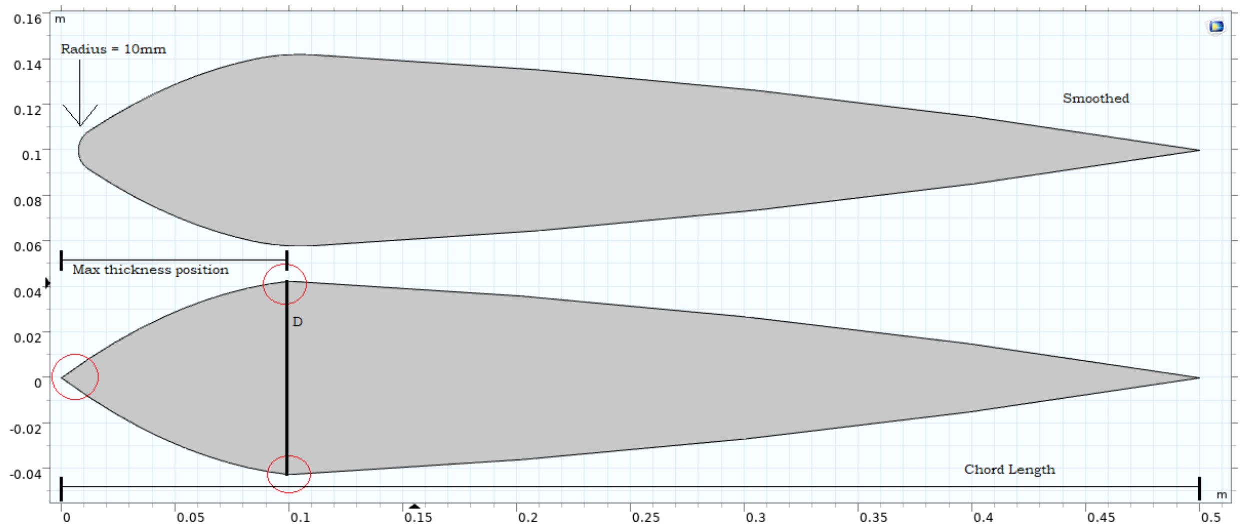


Figure 5.2. 1

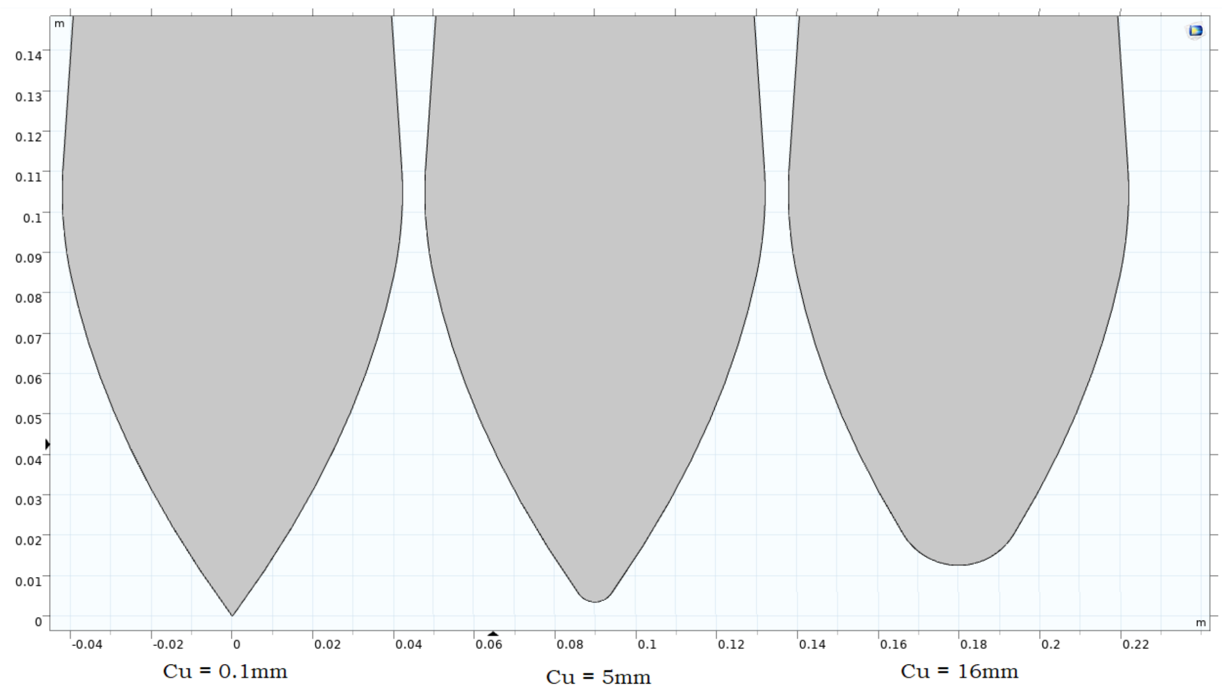


Figure 5.2. 2

<u>Dimensions</u>
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
D = 8.5cm
Max thickness position = 10cm
Cu (Radius of the Curvature of the tip of the nose) = (0.1-16.0) mm (varied)

With further decrease of Cu will not be properly meshed by the environment we use.

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0º to 30º with 5º step and Radius of the Curvature of the tip of the nose from 1mm to 16mm with step 3mm. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.

For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.2. 3:

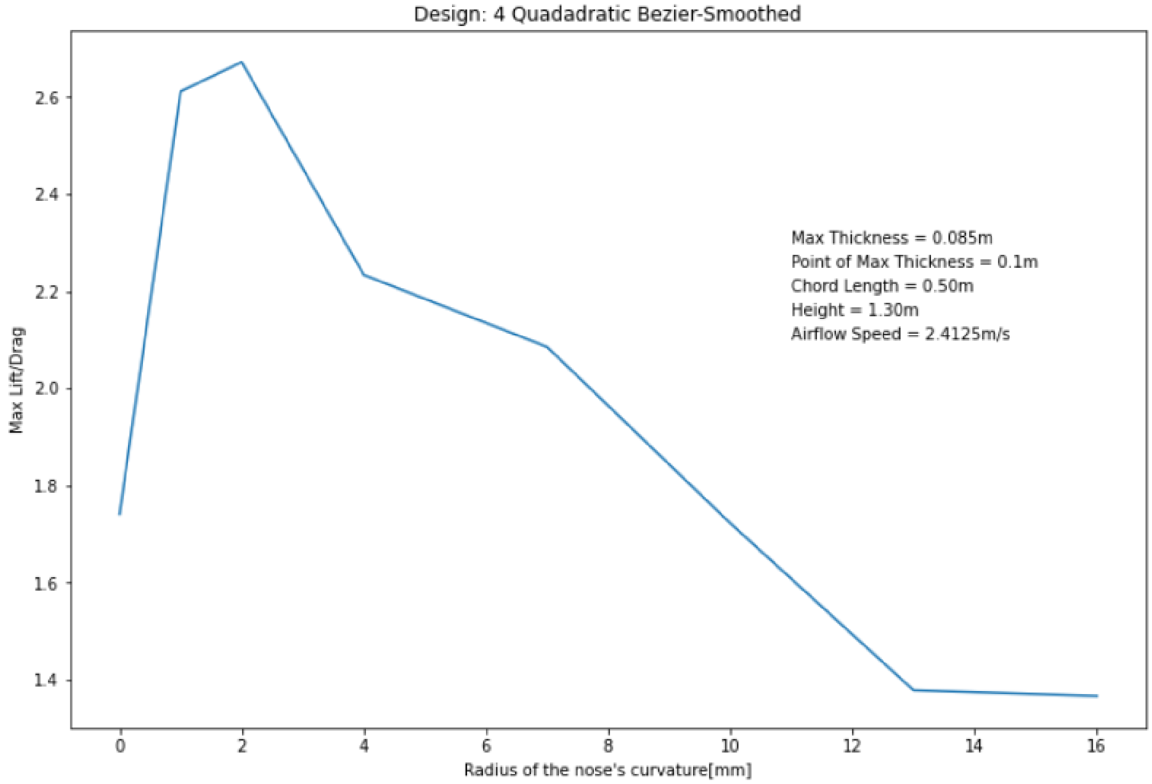


Figure 5.2. 3

The smaller the curvature of the nose's tip the greater the max Lift over Drag ratio. By removing completely, the curvature and using a point the Lift reduces significantly and so does the ratio.

At the following Figure 5.2. 4 we present the Lift and Drag Coefficient curves over the Angle of attack for curvature of the nose with Radius of 2mm:

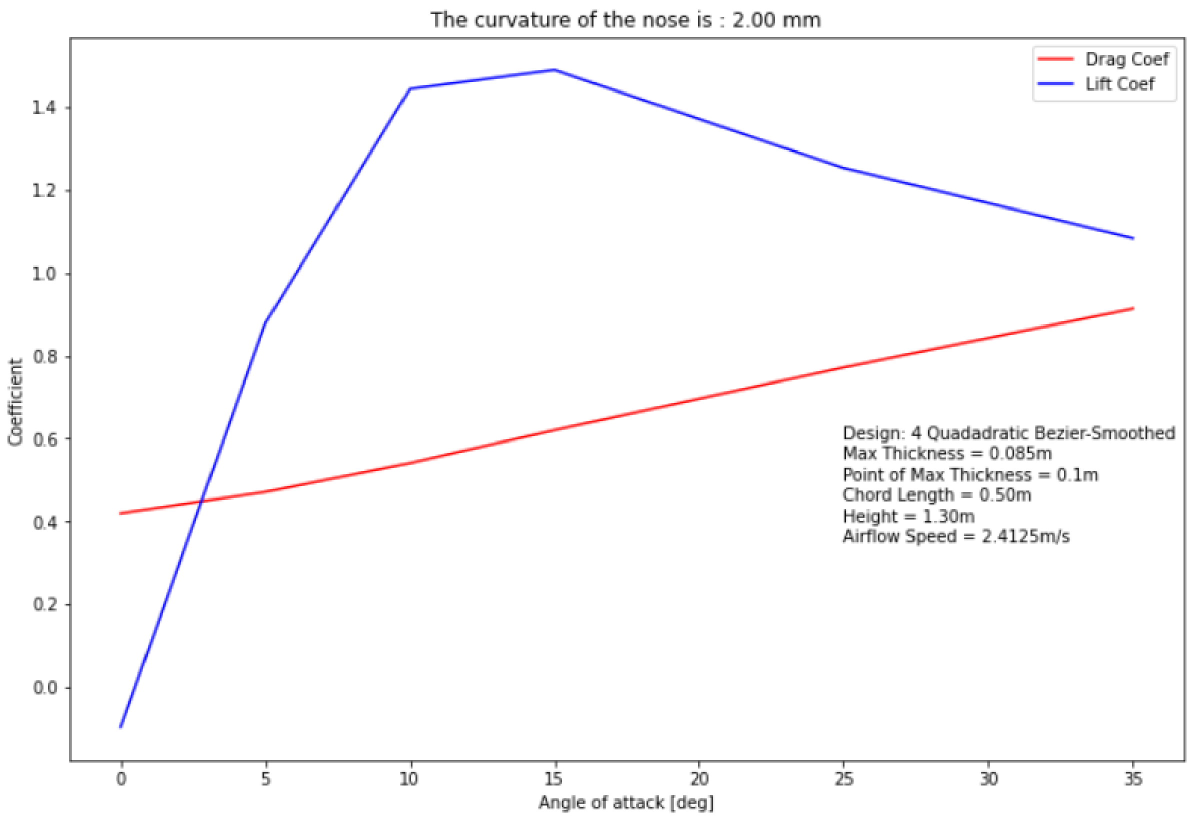

The curvature of the nose is : 2.00 mm
Drag Coef
Lift Coef
1.4
1.2
1.0
0.8
0.6
0.4
0.2
0.0
Coefficient
Design: 4 Quadadratic Bezier-Smoothed
Max Thickness = 0.085m
Point of Max Thickness = 0.1m
Chord Length = 0.50m
Height = 1.30m
Airflow Speed = 2.4125m/s
0
5
10
15
20
25
30
35
Angle of attack [deg]

Figure 5.2. 4

5.3 Nose Curvature investigation (y Direction)

Design

The design is the same as before with the difference that there isn't smoothening at the tip of the nose and the variable is the middle points of the two Quadratic Bezier that form the nose. Specifically, the variable displaces the middle point at the y direction (the direction of the chord) making the whole nose rounder like shown below at the Figure 5.3. 1:

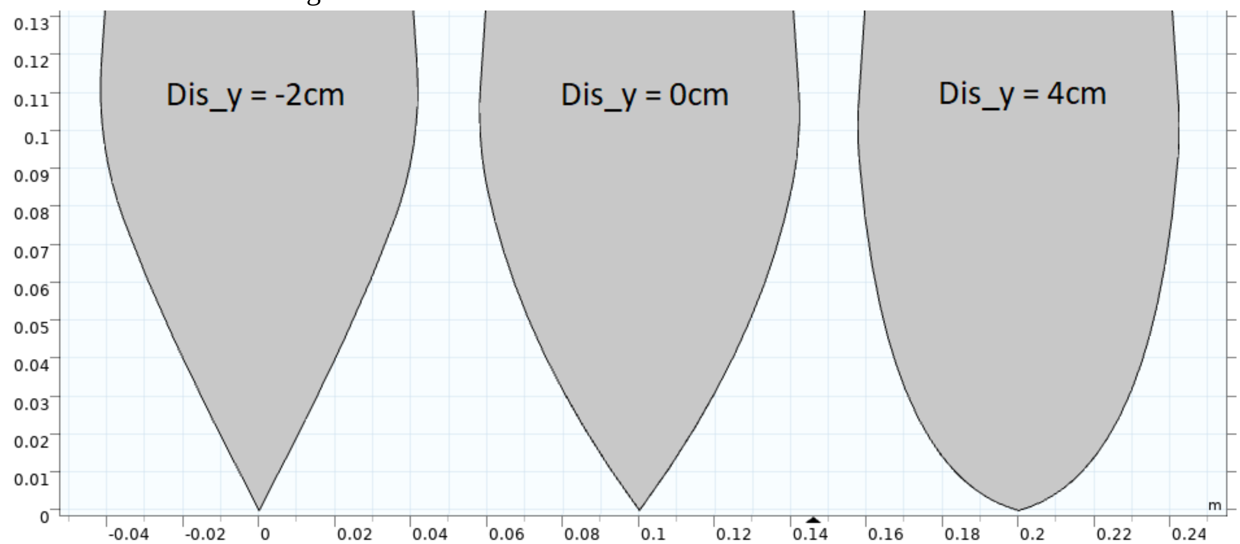


Figure 5.3. 1

<u>Dimensions</u>
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
D = 8.5cm
Max thickness position = 10cm
Dis_y (Displacement of the side at the y direction) = -2 to 4 cm

The positive values have the meaning that the direction of the displacement is outwards and the negative inwards.
With further decrease of dis_y the design starts to deform at its basic characteristics like the max thickness position and max thickness. This is a problem because we will see better results because the sail will be thinner and independently of the curvature of the nose.

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0^o to 30^o with 5^o step and Dis_y from -2cm to 4cm with step 1cm. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.
For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.3. 2:

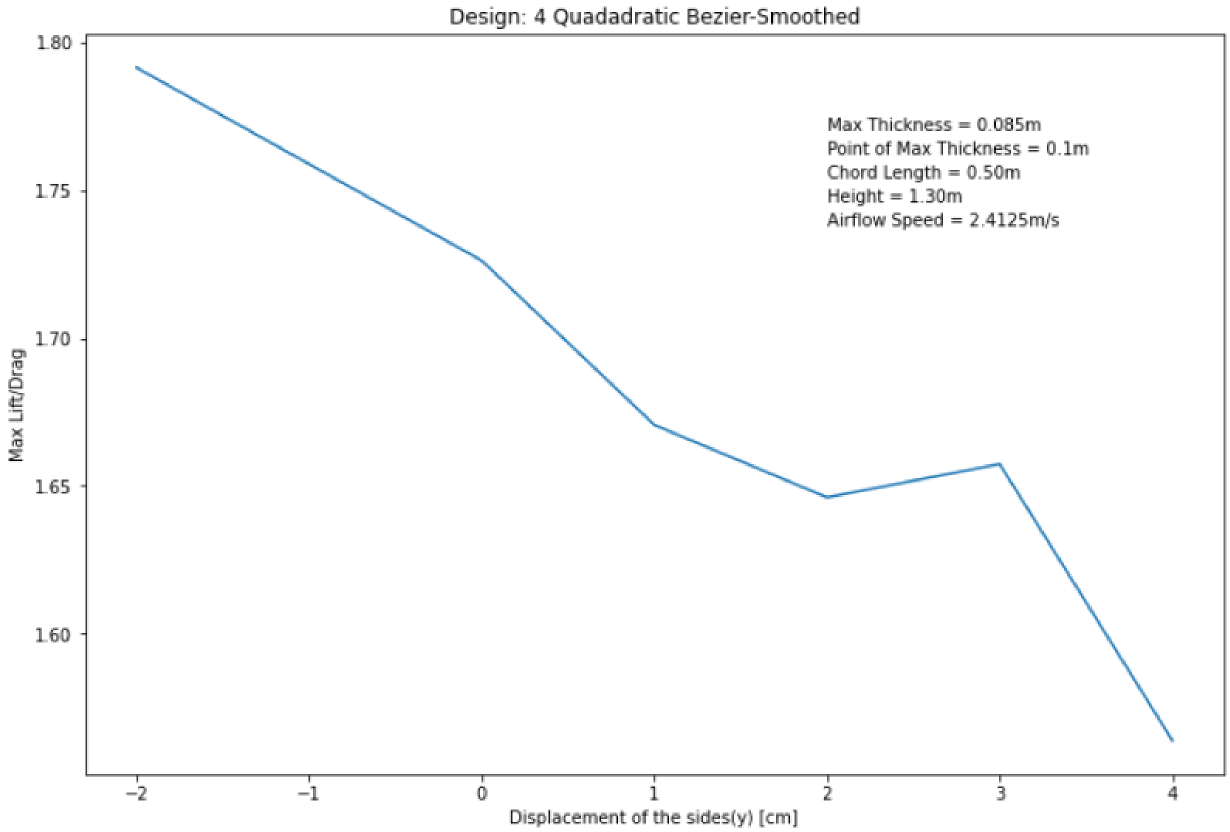


Figure 5.3. 2

As we can see as the more the inwards displacement at the y direction the better the Lift over Drag ratio.

5.4 Nose Curvature investigation (x Direction)

Design

We will use the previous design with Dis_y = 0 and not the better Dis_y = -2 as the combination with the displacement at the x direction we again have the problem of deformation at the basic characteristics of the sail like thickness and max thickness position. Here the middle points of the two Quadratic Bezier that form the nose are displaced at the x direction as shown at the Figure 5.4. 1.

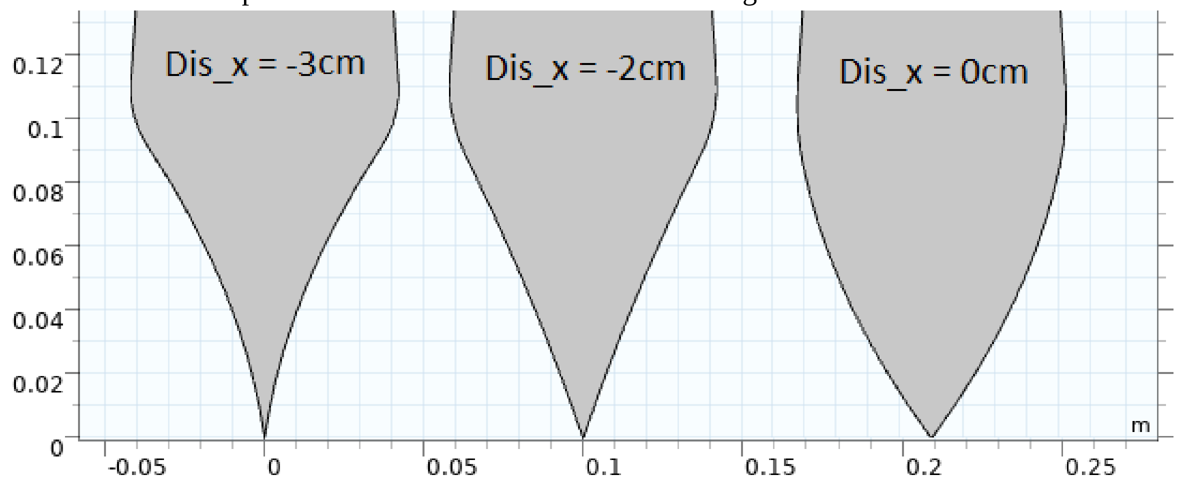


Figure 5.4. 1

<u>Dimensions</u>
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
D = 8.5cm
Max thickness position = 10cm
Dis_y = 0 cm
Dis_x (Displacement of the side at the x direction) = -3 to 0 cm

As before the positive values has the meaning that the direction of the displacement is outwards and the negative inwards.

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0^{o} to 30^{o} with 5^{o} step and Dis_x from -3cm to 0cm with step 1cm. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.
For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.4. 2:

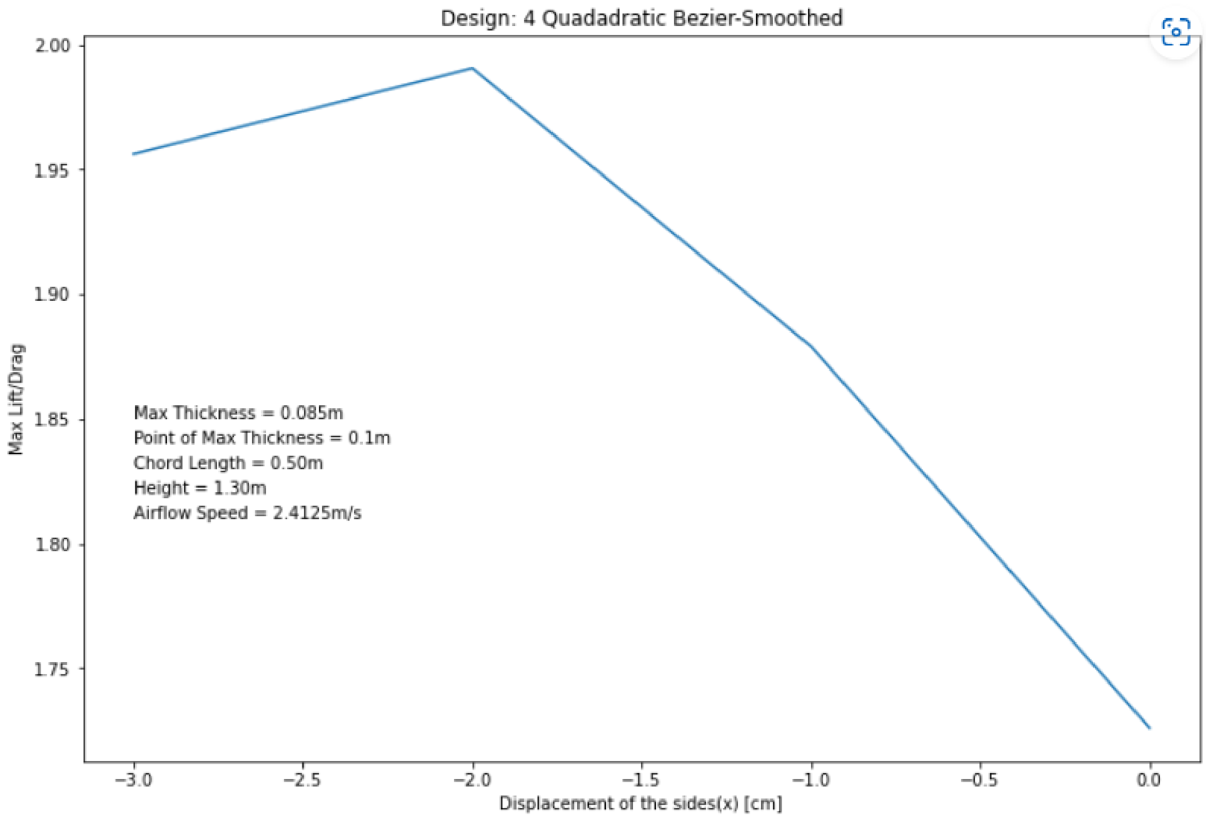


Figure 5.4. 2

As we can see as the more the inwards displacement at the x direction the better the Lift over Drag ratio, until we reach the threshold at Dis_x = -2cm where the nose of the sail becomes to narrow.
Below we have the Figure 5.4. 3 we have the Lift and Drag Coefficients over the Angle of attack using Dis_y = 0cm and Dis_x = -2cm.

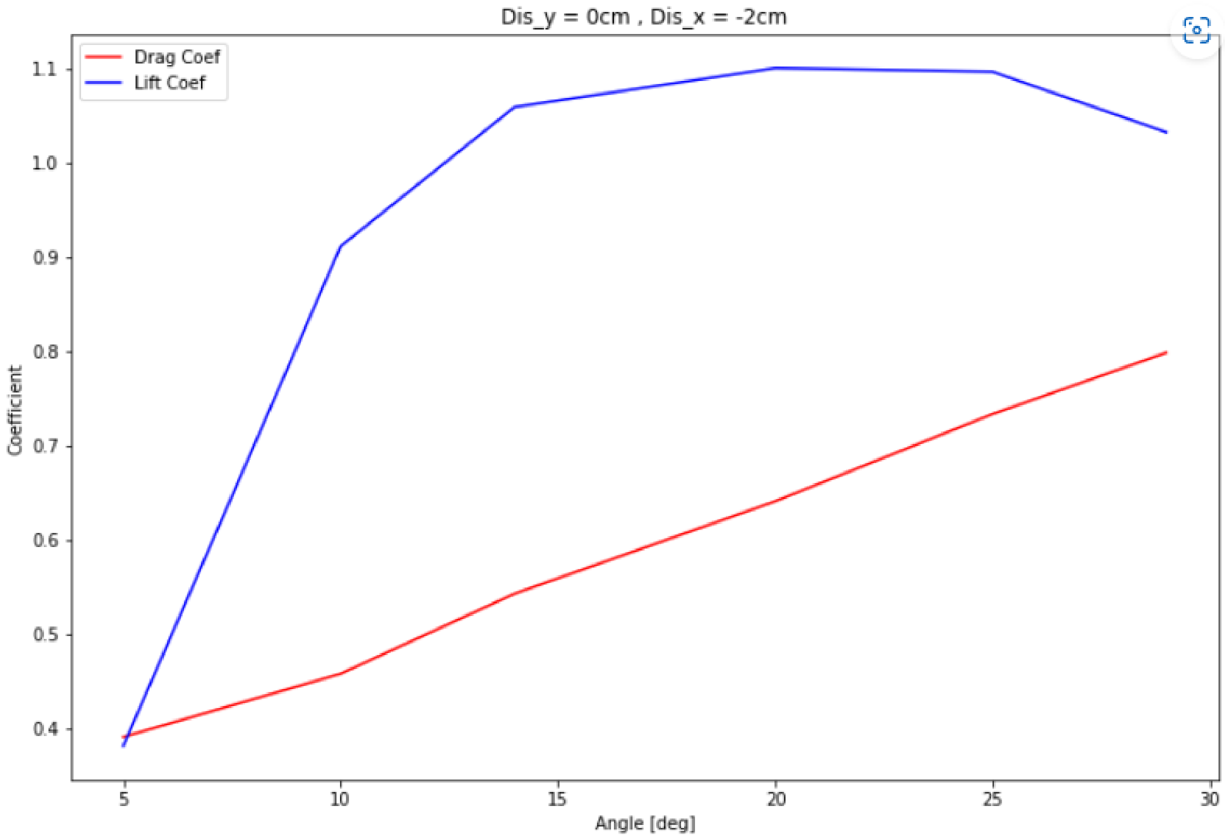

Dis_y = 0cm , Dis_x = -2cm
Drag Coef
Lift Coef
1.1
1.0
0.9
0.8
0.7
0.6
0.5
0.4
Coefficient
5
10
15
20
25
30
Angle [deg]

Figure 5.4. 3

5.5 Camber investigation

Design

We use the same design as before with Displacements of the nose's sides Dis_y = 0cm and Dis_x = -2cm and curvature of the nose's tip of Cu = 2mm. Now we will investigate the increase of the camber that bends the sail at one direction as shown below at the Figure 5.5. 1:

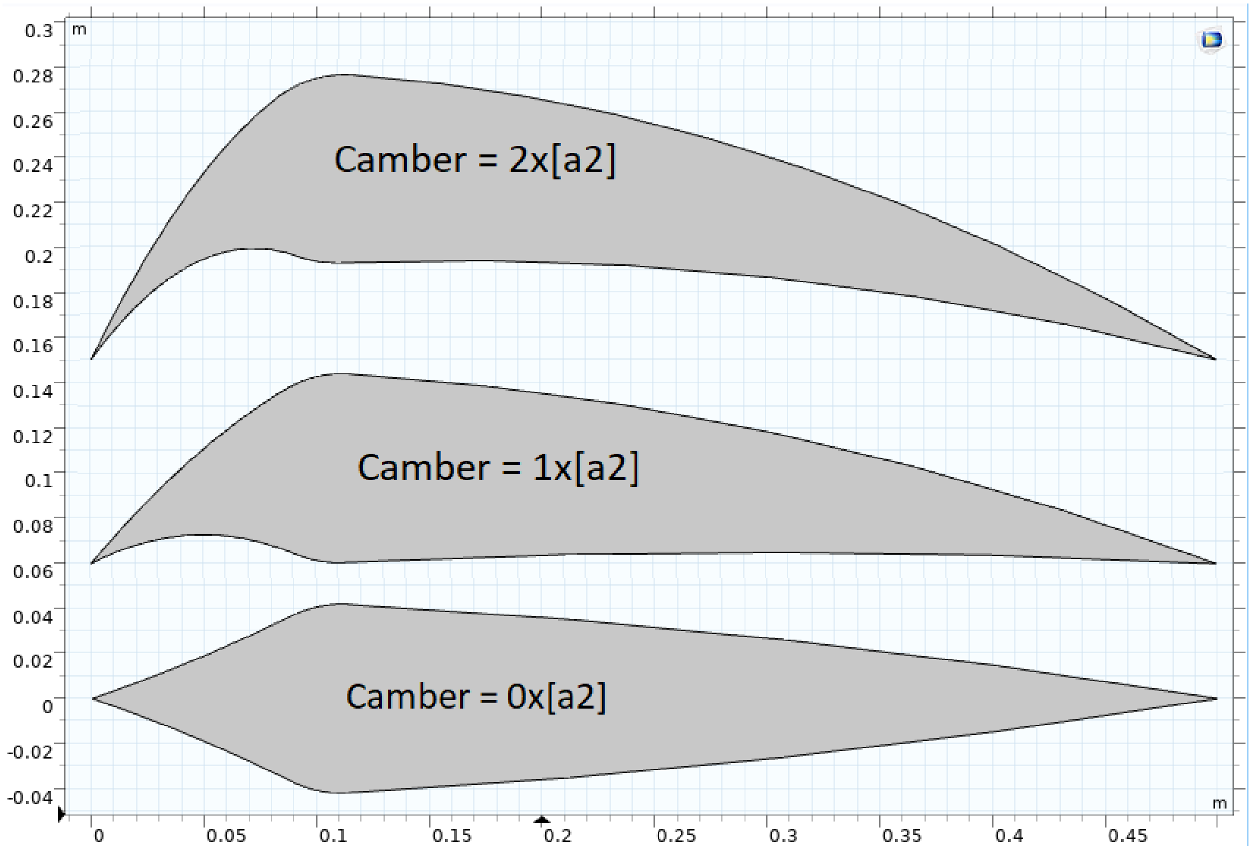


Figure 5.5. 1

<u>Dimensions</u>

Width (chord length) = 0.5 m

Airfoil Length = 1.30 m

Thickness = 8.5cm

Max thickness position = 10cm

Cu = 1mm

Dis_y = 0cm

Dis_x = -2cm

Camber = 0 to 2x[a2] with step 0.5x[a2] where a2 is the radius of the imaginary circle at the point of max thickness: a2 = 4.25cm

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0º to 30º with 5º step and Camber from 0x[a2] to 2x[a2] with step 0,5x[a2]. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.
For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.5. 2:

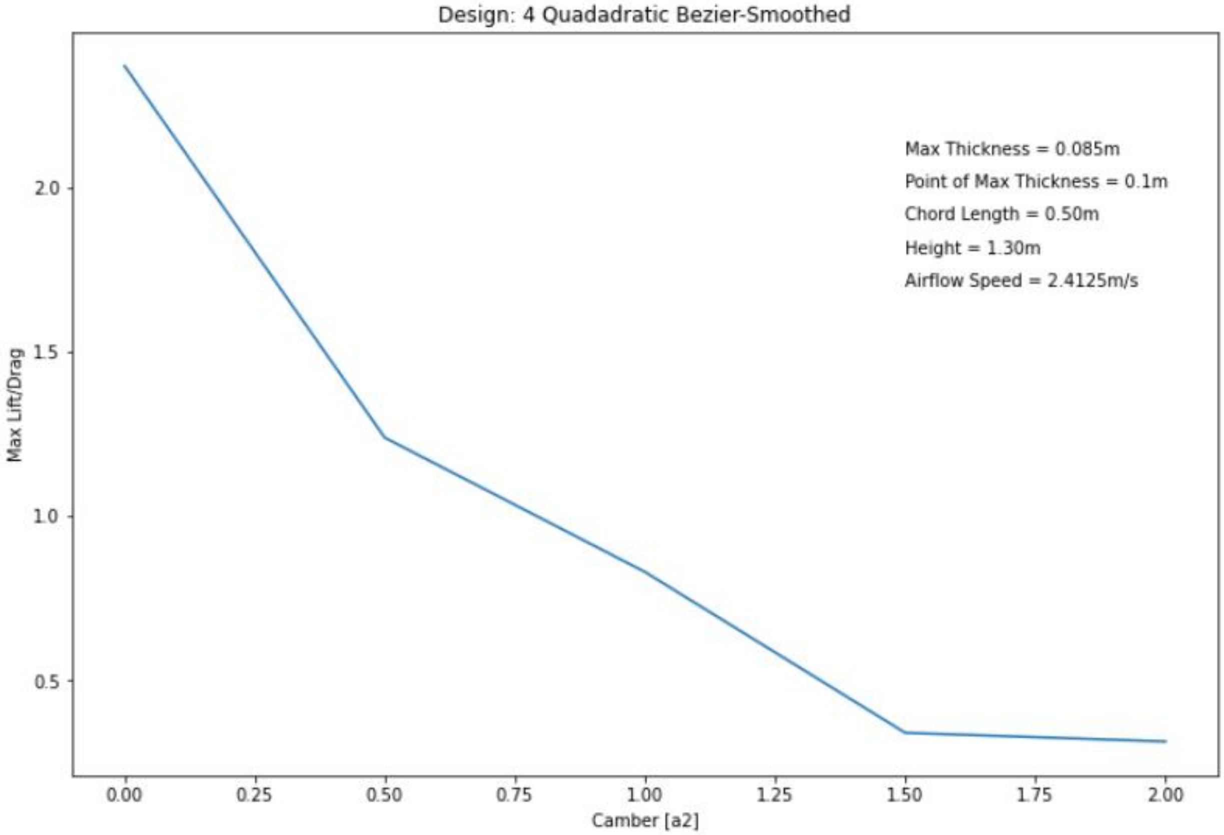


Figure 5.5. 2

Increasing the camber reduces the Lift over Drag ratio so we will use Camber = 0.
Below we have the graph of Lift and Drag Coefficients over the Angle of attack Camber = 0.

This result is worse than the one without the displacement optimization at the Curvature of the nose’s tip investigation so we will run again the last investigations but with Cu = 2mm

5.6 Nose Curvature investigation (y Direction) with Cu = 2mm

Design

We use the same design as the one at *3.4_Nose Curvature investigation (y Direction)* but with curvature of the nose's tip Cu = 2mm. Here, as before, the middle points of the two Quadratic Bezier that form the nose are displaced at the y direction as shown at the Figure 5.6. 1.

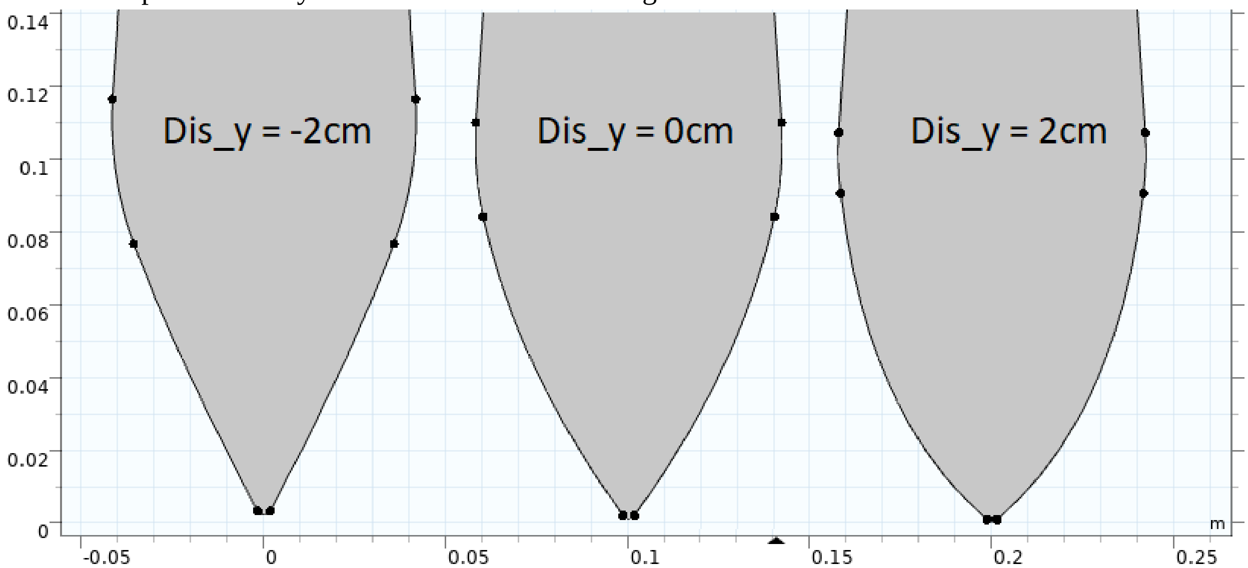


Figure 5.6. 1

Dimensions
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
D = 8.5cm
Max thickness position = 10cm
Cu = 2mm
Dis_y (Displacement of the side at the y direction) = -2 to 2 cm

The positive values have the meaning that the direction of the displacement is outwards and the negative inwards.
With further decrease of Dis_y the design starts to deform at its basic characteristics like the max thickness position and max thickness. This is a problem because we will see better results because the sail will be thinner and independently of the curvature of the nose.

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0^o to 35^o with 5^o step and Dis_y from -2cm to 2cm with step 1cm. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.
For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.6. 2:

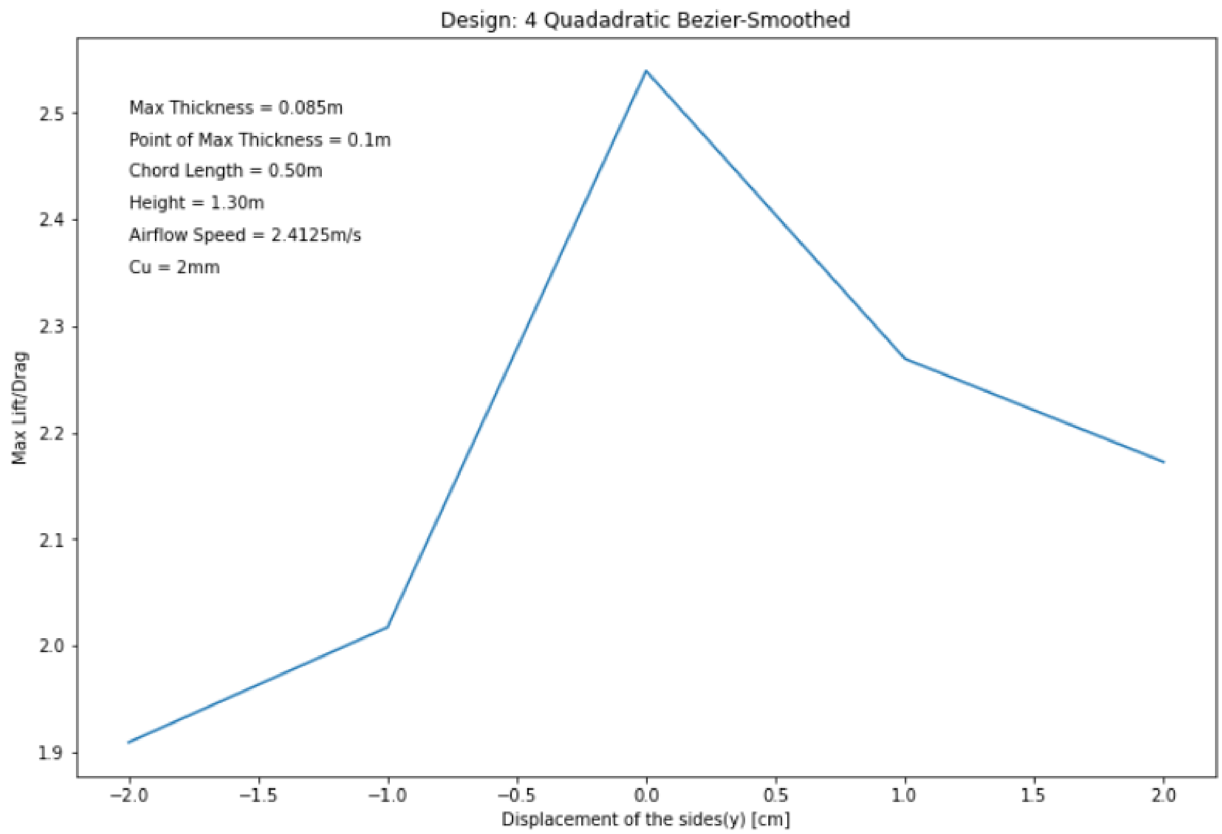


Figure 5.6. 2

As we can see the displacement at the y direction does not help.

5.7 Nose Curvature investigation (x Direction) with Cu = 2mm

Design

We use the same design as the one at *3.5_Nose Curvature investigation (x Direction)* but with curvature of the nose's tip Cu = 2mm. Here, as before, the middle points of the two Quadratic Bezier that form the nose are displaced at the x direction as shown at the Figure 5.7. 1.

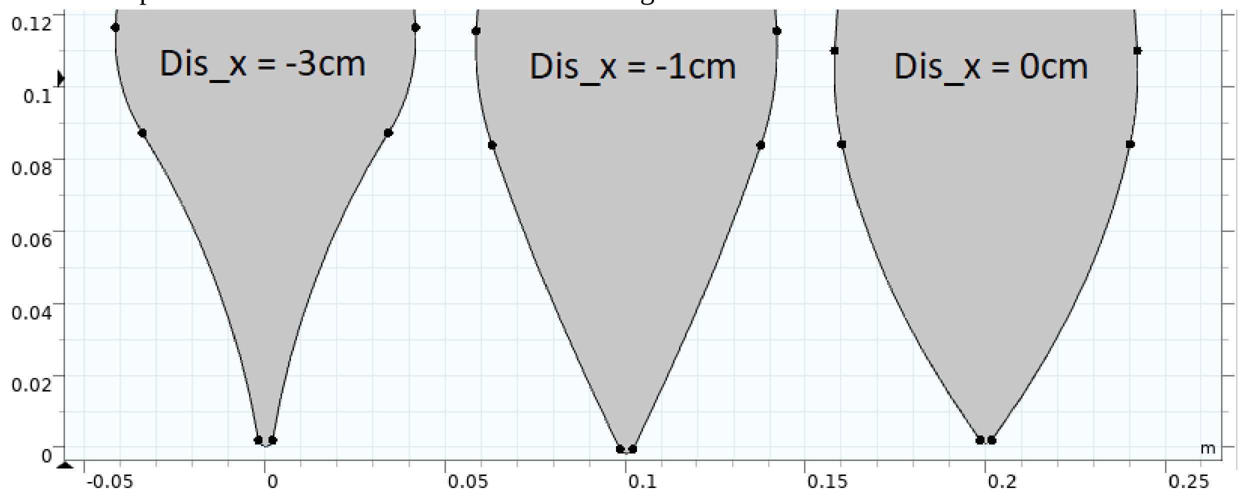


Figure 5.7. 1

Dimensions
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
D = 8.5cm
Max thickness position = 10cm
Cu = 2mm
Dis_y = 0 cm
Dis_x (Displacement of the side at the x direction) = -3 to 0 cm

As before the positive values has the meaning that the direction of the displacement is outwards and the negative inwards.

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0^o to 30^o with 5^o step and Dis_x from -3cm to 0cm with step 1cm. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.
For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.7. 2:

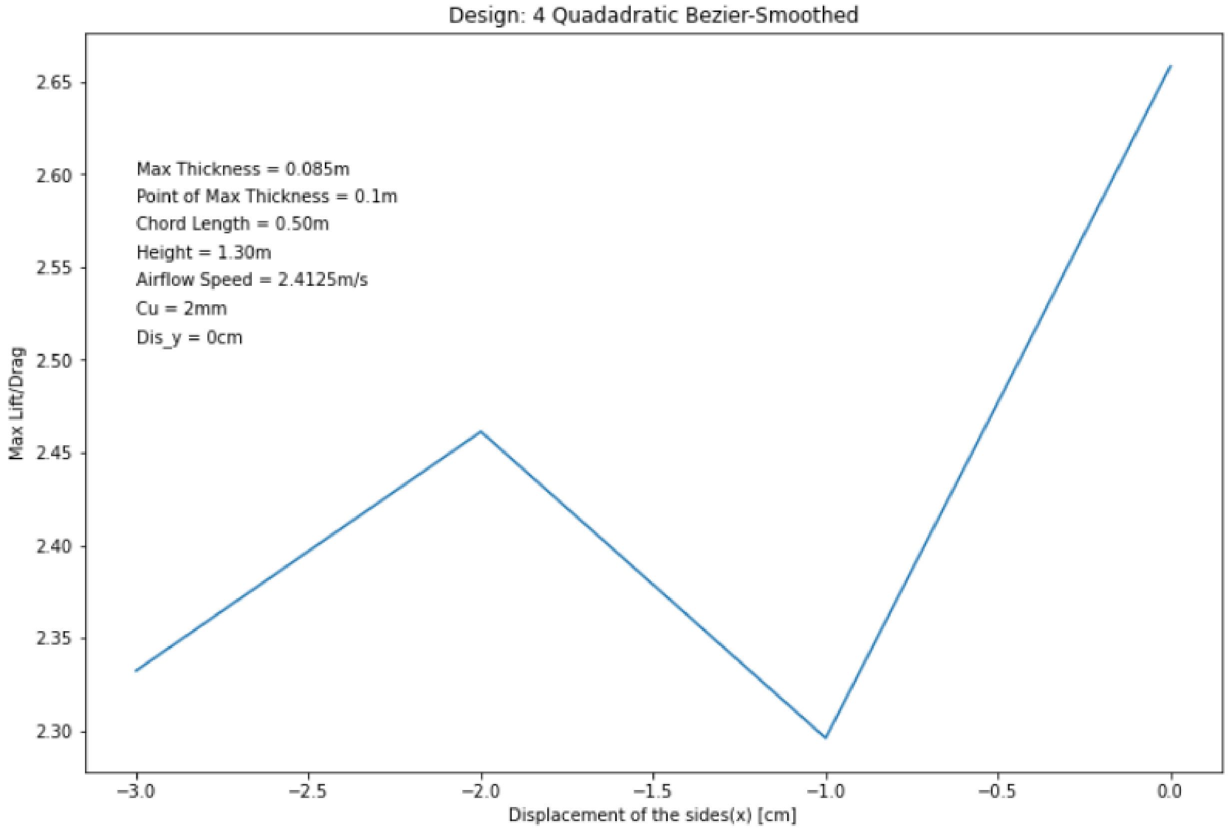


Figure 5.7. 2

As we can see the displacement at the y direction does not help.

5.8 Camber investigation with Cu = 2mm

Design

We use the same design as the one at *3.6_Camber investigation with Cu = 2mm* but without the displacement of the nose's sides. Some of the Camber values are displaced at the Figure 5.8. 1

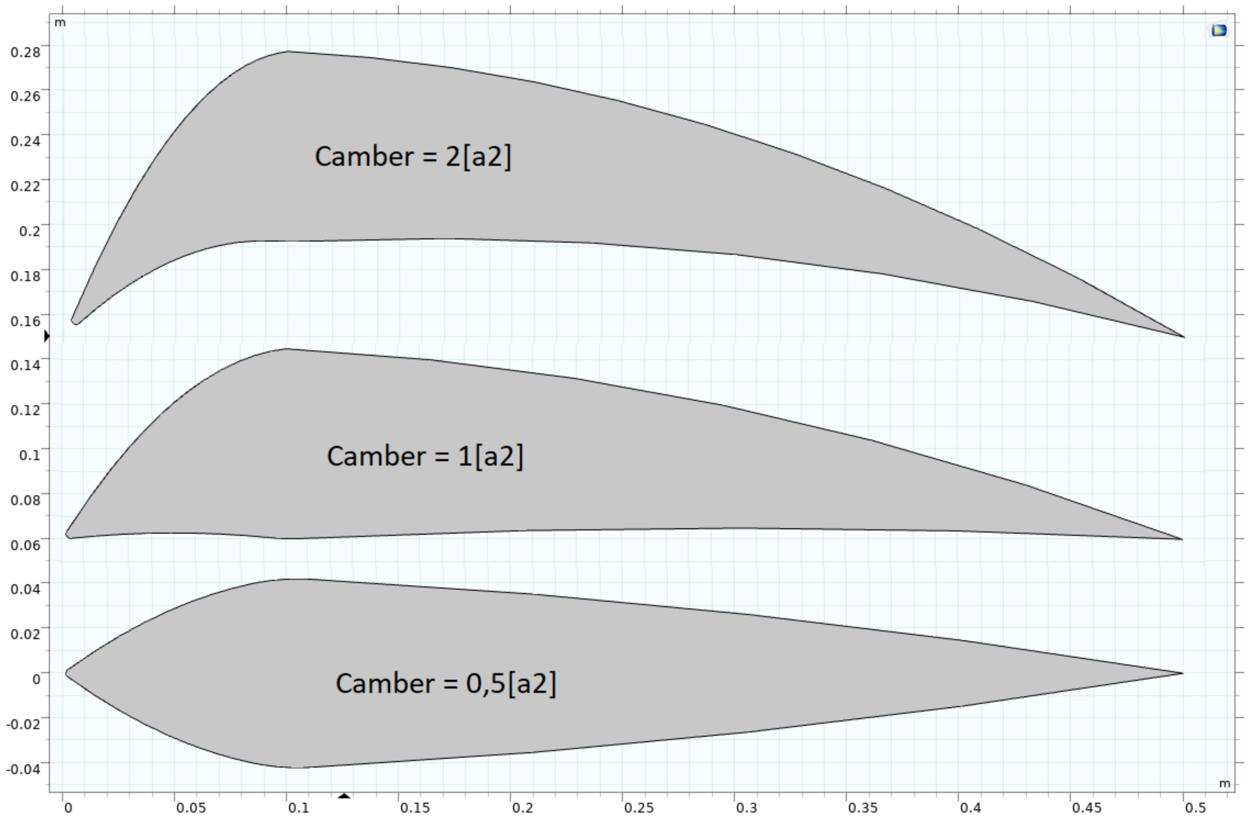


Figure 5.8. 1

Dimensions
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
Thickness = 8.5cm
Max thickness position = 10cm
Cu = 1mm
Dis_y = 0cm
Dis_x = -2cm
Camber = 0 to 2x[a2] with step 0.5x[a2] where a2 is the radius of the imaginary circle at the point of max thickness: a2 = 4.25cm

Results

The sail is symmetric so the angles of attack we will use will be only from one direction. We used from 0º to 30º with 5º step and Camber from 0x[a2] to 2x[a2] with step 0,5x[a2]. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.
For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 8.8. 2:

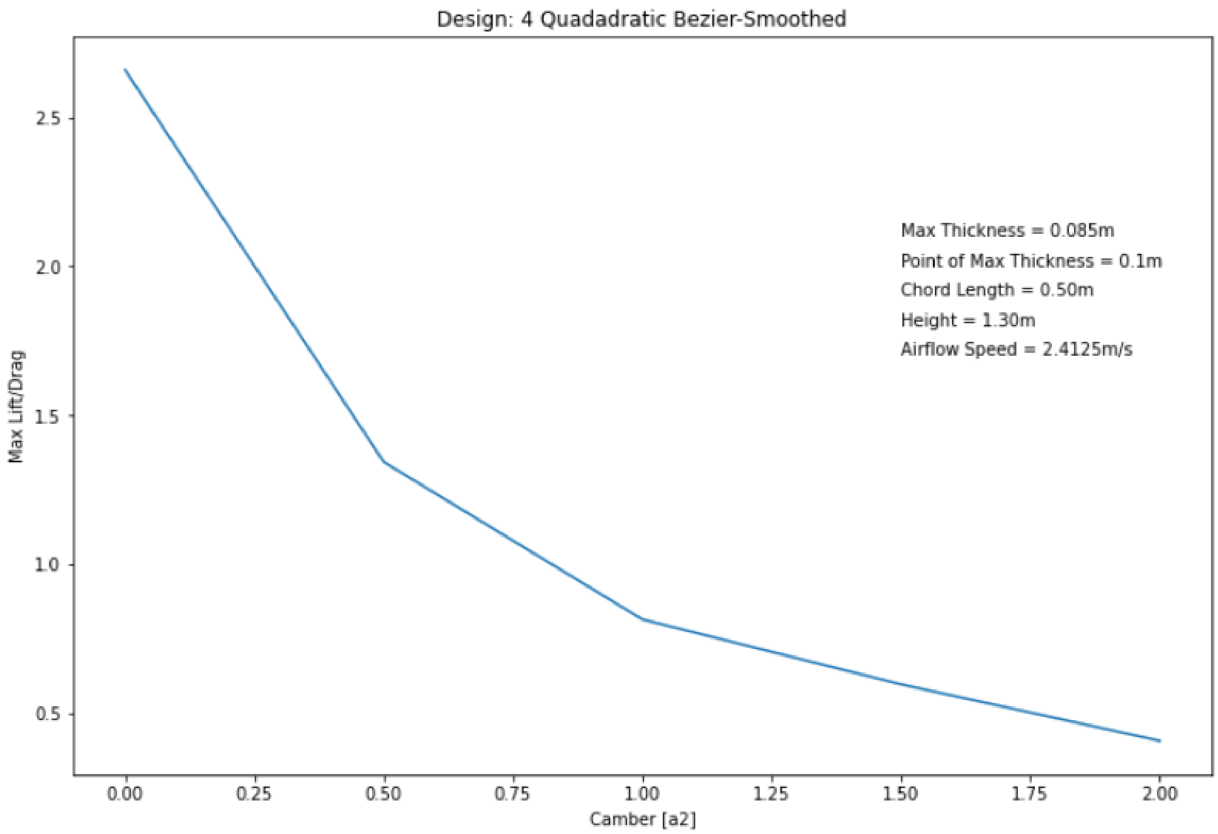


Figure 5.8. 2

Increasing the camber reduces the Lift over Drag ratio so we will use Camber = 0.

5.9 Two-Part sail distance investigation

Design

Duplicating the sail from the previous investigation with Camber = 0 we have the following system of airfoils. The variable Gap is the distance we move them apart as shown at the Figure 5.9. 1

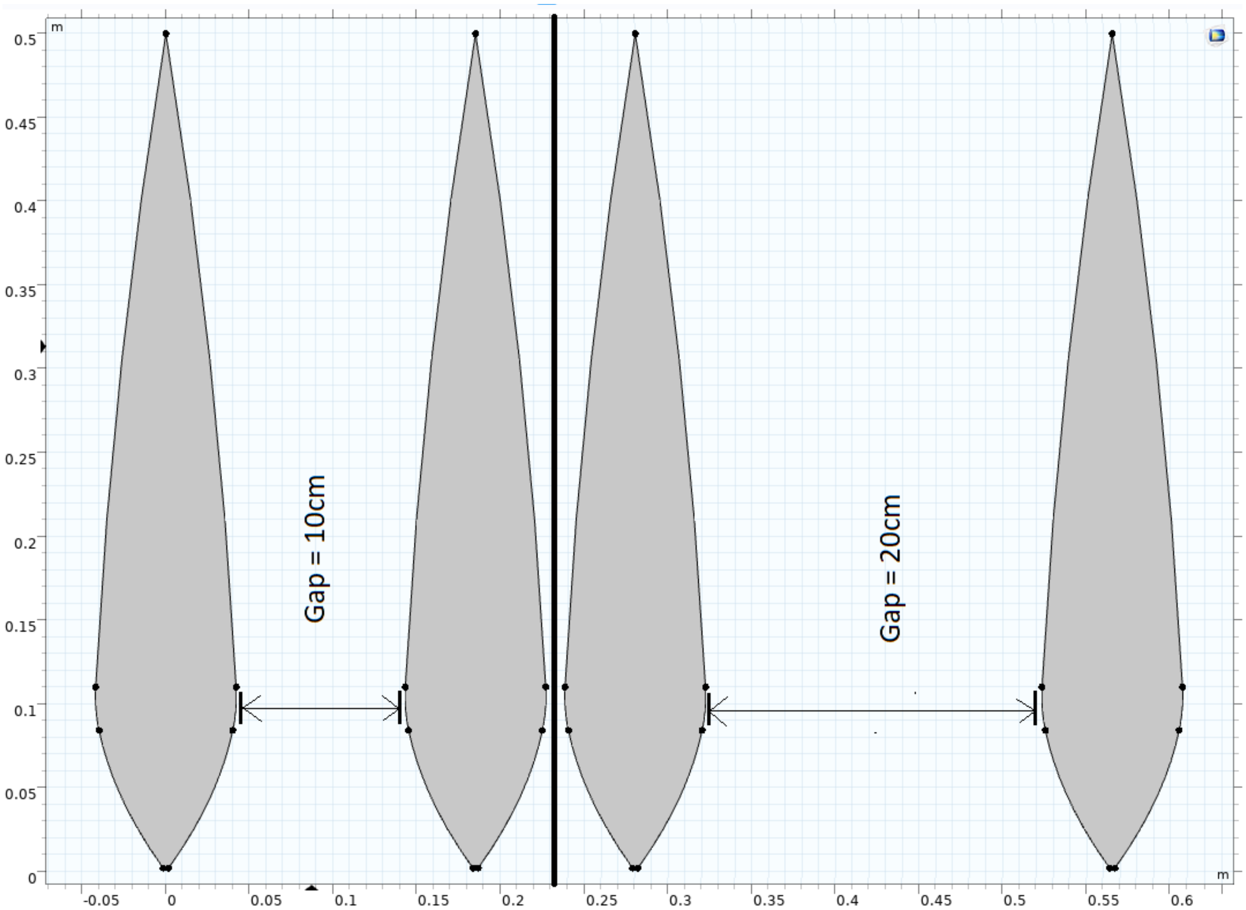


Figure 5.9. 1

<u>Dimensions</u>
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
Thickness = 8.5cm
Max thickness position = 10cm
Cu = 2mm
Dis_y = 0cm
Dis_x = -2cm
Camber = 0
Gap = 10-80 cm

We will increase the Width of the block by 1m (2*Chord length) to compensate for the second sail.

Results

The system of sails is symmetric so the angles of attack we will use will be only from one direction. We used from 0º to 35º with 5º step and Gap from 10 cm to 90 cm with step 10 cm. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle. For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.9. 2:

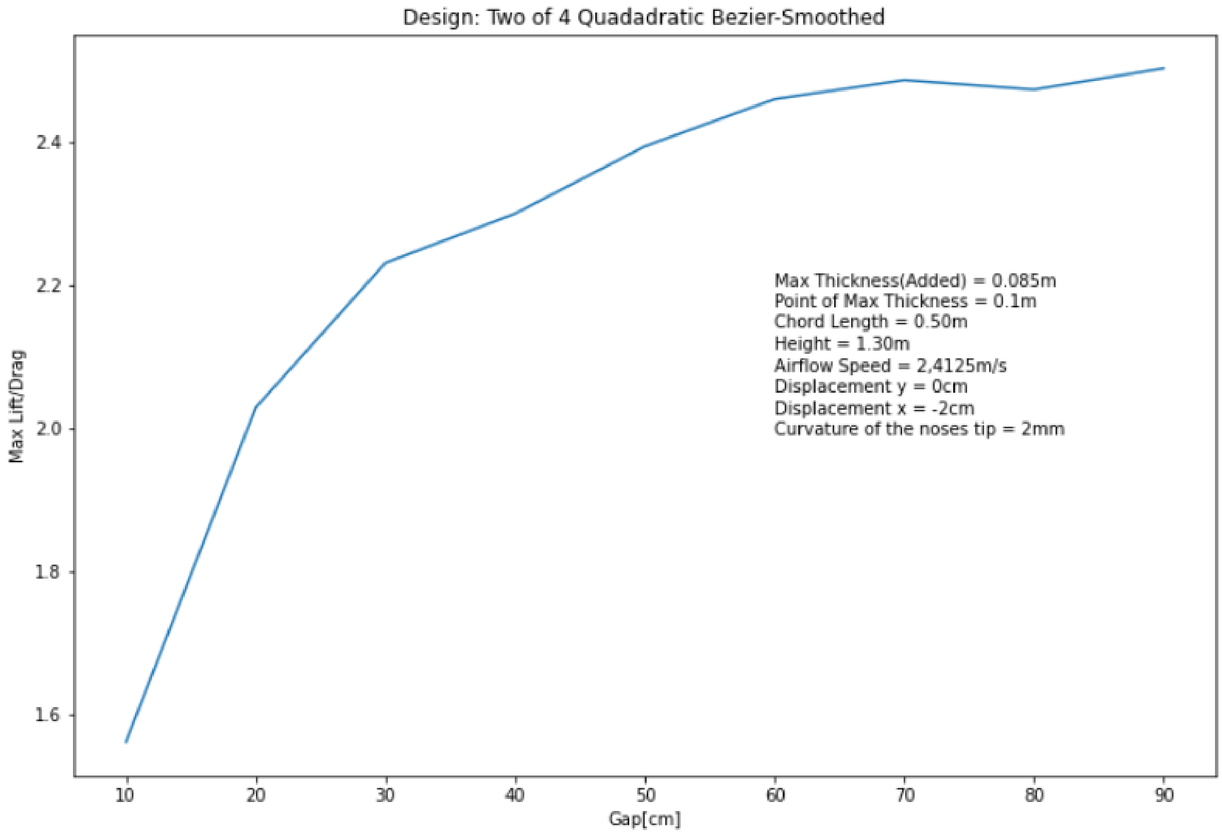


Figure 5.9. 2

The bigger the gap the better the Lift over Drag ratio. It seems like the interference of the two airfoils is not beneficial. Adding a second sail reduces the Lift over Drag ratio

5.10 Two-Part sail between Angle investigation

Design

We have rotated the one sail in relation to the other. The distance between the sails at the max thickness position is 60cm as shown at the Figure 5.10. 1.

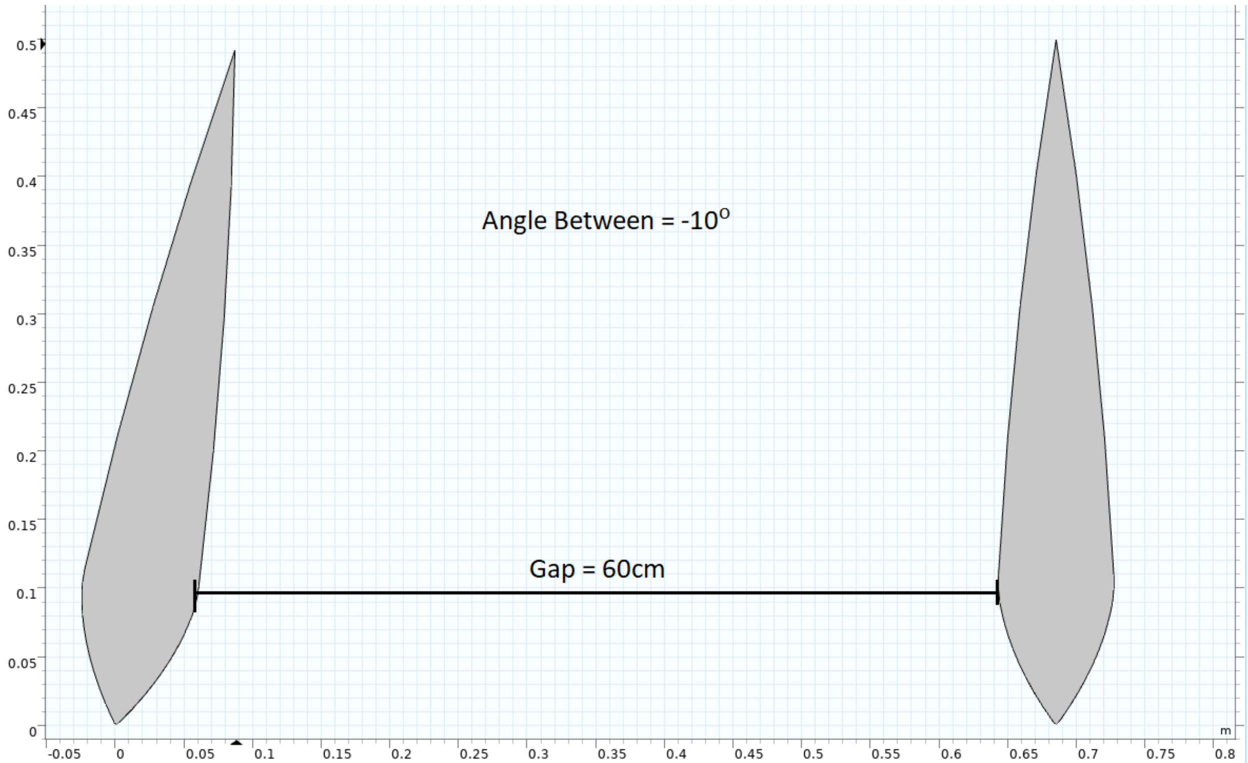


Figure 5.10. 1

<u>Dimensions</u>
Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
Thickness = 8.5cm
Max thickness position = 10cm
Cu = 2mm
Dis_y = 0cm
Dis_x = -2cm
Camber = 0
Gap = 60cm
The rotation axis distance from nose = 8,5cm
B_Angle (Angle Between the sails) = -10^{o} to 10^{o}

Results

The system of sails is symmetric but the axis of symmetry changes. To compensate we will used Angles from -10º to 45º with 5º step and B_Angle from -10 º to 10º with step 5º. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.

For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.10. 2:

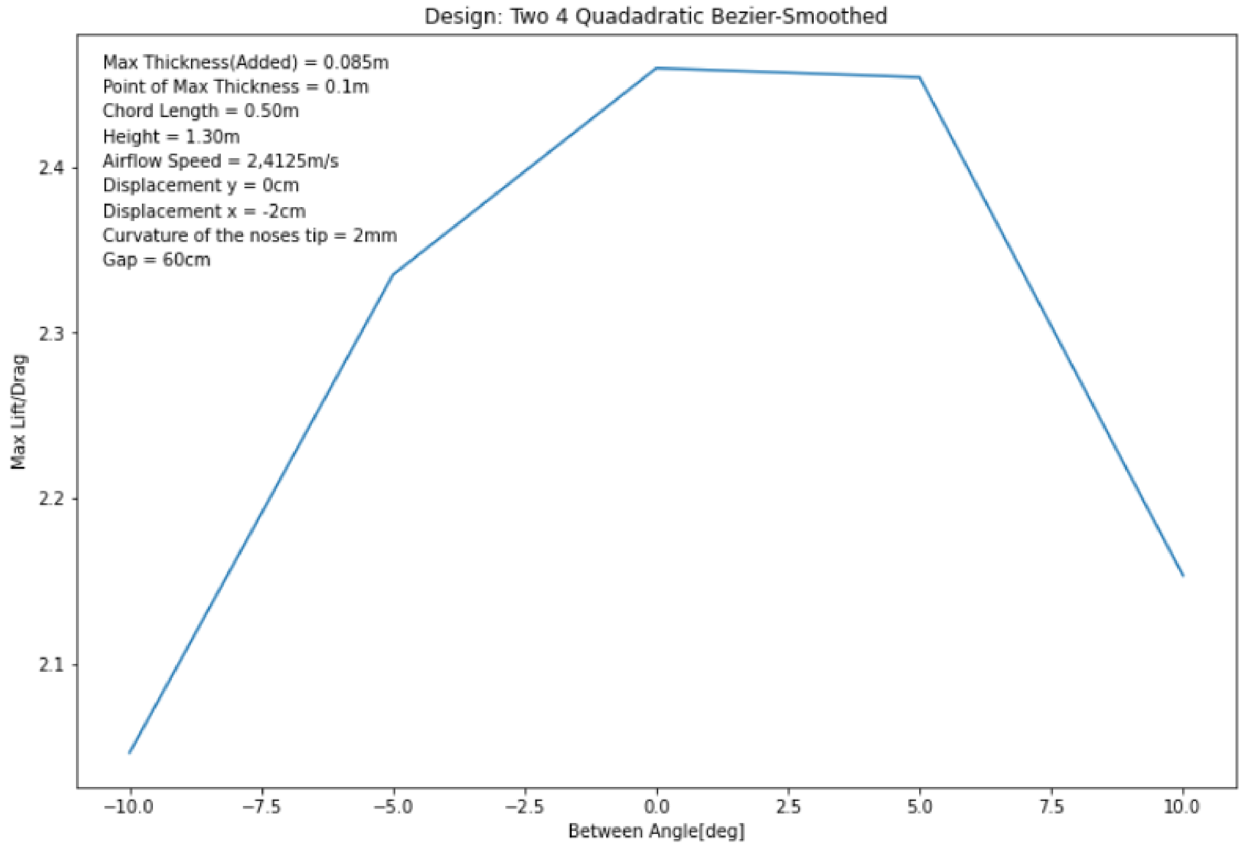


Figure 5.10. 2

Rotating the one sail in relation to the other reduces the Lift over Drag ratio.

5.11 Two-Part sail Camber investigation

Design

We have added opposite camber to the sails. The distance between the sails at the max thickness position is 60cm as shown at the Figure 5.11. 1.

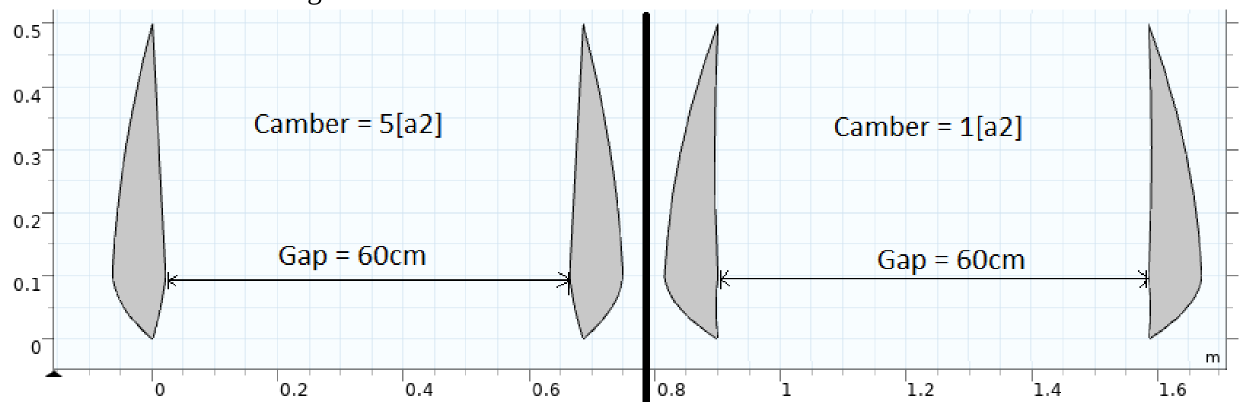


Figure 5.11. 1

<u>Dimensions</u>

Width (chord length) = 0.5 m
Airfoil Length = 1.30 m
Thickness = 8.5cm
Max thickness position = 10cm
Cu = 2mm
Dis_y = 0cm
Dis_x = -2cm
Gap = 60cm
Camber = 0 to 2 [a2]

Results

The system of sails is symmetric so the angles of attack we will use will be only from one direction. We used from 0º to 35º with 5º step and Camber from 0x[a2] to 2x[a2] with step 0,5x[a2]. The metric for our results will be the Lift over Drag value that is the maximum for each variation of the radius of the small circle.

For Reynolds number value 82000 and wind speed 2,4125 m/s we have the following results portrait at Figure 5.11. 2:

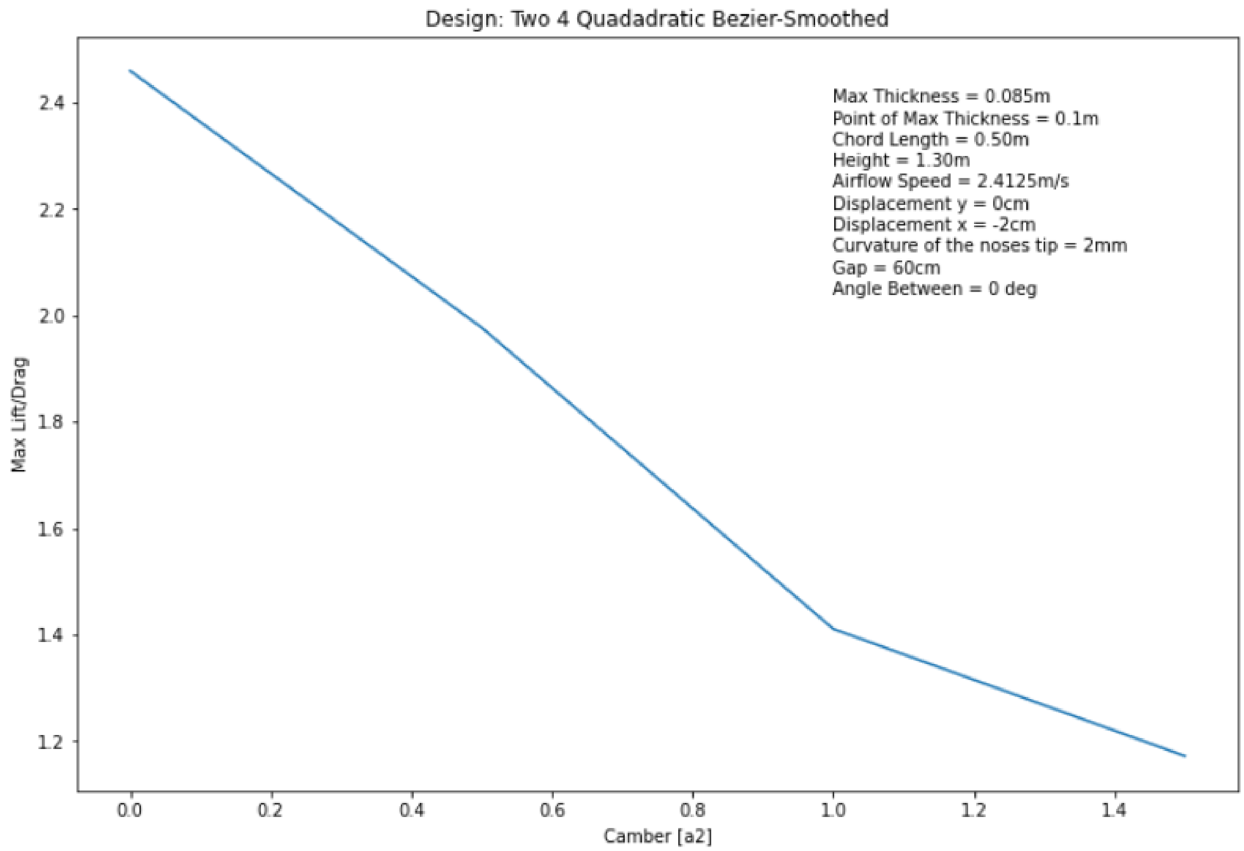


Figure 5.11. 2

Adding camber reduces the Lift over Drag ratio.

5.12 Comparison of the one and two sail system designs

We will compare the Lift and Drag coefficients data for the one and the two sail system designs. The Design is from the 5.2 investigation with the curvature of the nose's tip with Radius 2mm.

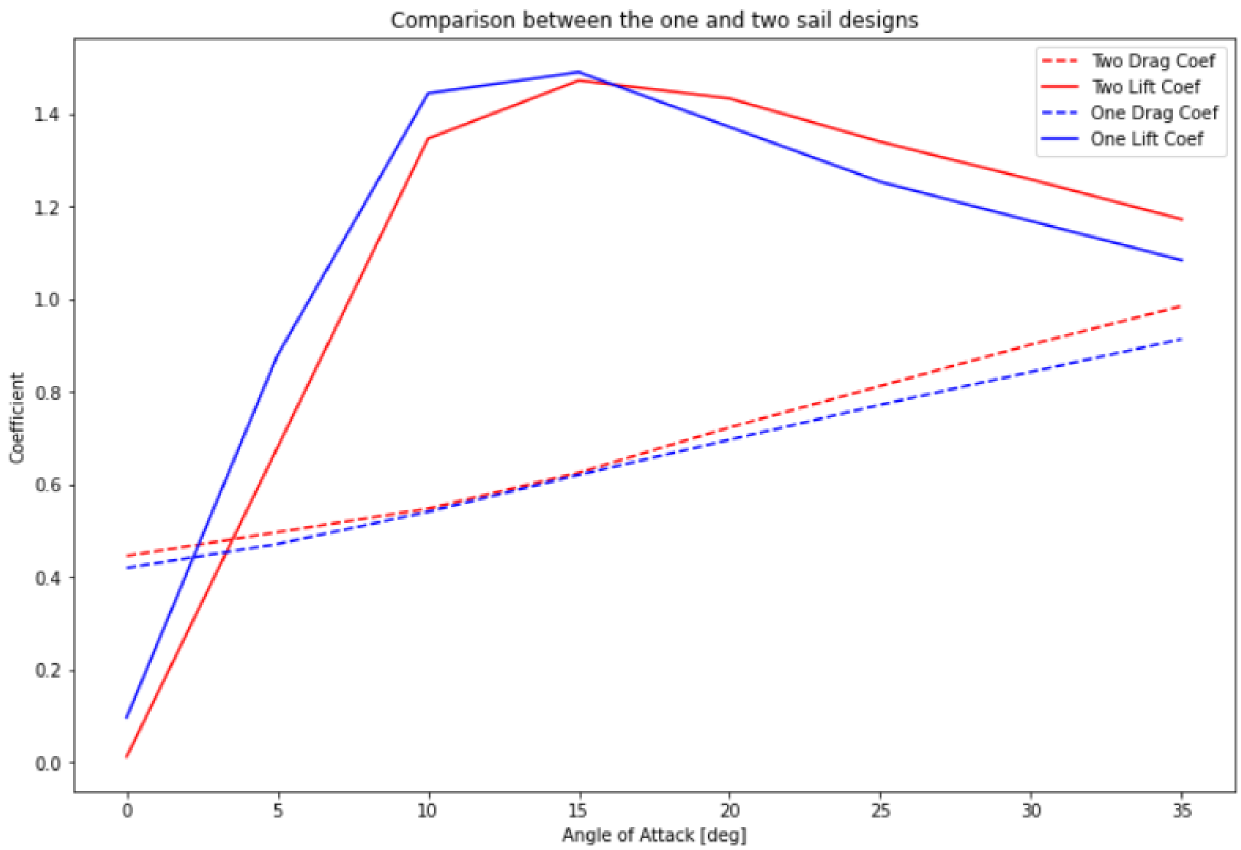


Figure 5.12. 1

As we can see the curves are pretty similar.

5.13 Conclusions

- The smaller the thickness the better the Lift over Drag ratio.
- The smaller the curvature of the nose's tip the greater the max Lift over Drag ratio.
- The inwards displacement at the x direction the better the Lift over Drag ratio.
- Increasing the camber reduces the Lift over Drag ratio.
- The bigger the gap the better the Lift over Drag ratio but worse than the one sail case. The interference of the two airfoils is not beneficial.
- Rotating the one sail in relation to the other reduces the Lift over Drag ratio.

6 Optimal Design

6.1 Design Presentation

The optimal design is portraited with a 2D representation at the Figure 6.1

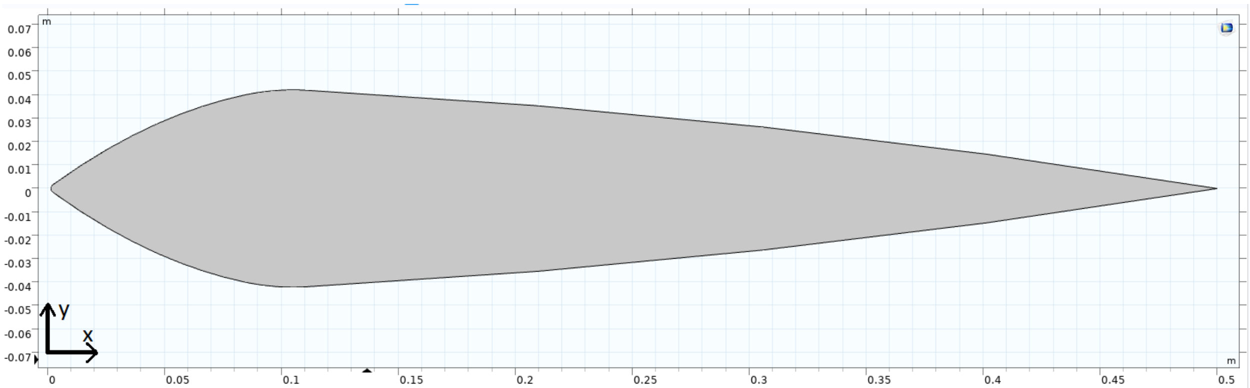


Figure 6. 1

<u>Design Characteristics:</u>
Height = 1,3m
Width = 0.5m
Max Thickness = 8,5cm
Max Thickness position from the nose's tip = 10cm
Radius of the imaginary circle at the point of max thickness = 4,25cm
The design is being extruded to a 3d object.

6.2 Model Instructions

A model for replicating the results for this design can be built with the instructions presented below:

New

In the **New** window, click **Model Wizard**.

Model Wizard

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow> Single-Phase Flow>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

Global Definitions

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
rho_Air	1.225[kg/m^3]	1.225 kg/m^3	Air Desity
mu_Air	1.802E-5[Pa*s]	1.802E-5 Pa*s	Air viscosity coefficient
Re	82000	82000	Reynolds number
V_Air	Re*mu_Air/(rho_Air*L)	2.4125 m/s	Airflow speed

Sail Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
2 At the **Label** text field, type Sail Parameters.
3 In the **Settings** window for **Parameters**, locate the **Parameters** section.
4 In the table, enter the following settings:

Name	Expression	Value	Description
L	0.5[m]	0.5 m	Characteristic Length
Length	1.3[m]	1.3 m	Sail Length
Width	L	0.5 m	Sail Width
D	0.085[m]	0.085 m	Diameter
MaxTh	0.1[m]	0.1 m	Max Thickness position
a	D/2	0.0425 m	Big circle middle
Cu	2	2	Curvature of the nose
Angle	15[deg]	0.2618 rad	Angle of attack

Geometry 1

1 At the **Graphics** window click at the **Transparency** button.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the text fields, enter the following settings:

Width	20*L	**m**
Depth	8*L	**m**
Height	8*L	**m**

4 Locate the **Position** section. At the **Base** list, choose **Center**.
5 In the text fields that follows enter the following settings:

x	13*L/2	**m**
y	0	**m**
z	Length/2	**m**

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Model Builder** window, under Work Plane 1, click Plane Geometry.

Down Nose (qb1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Quadratic Bezier**.
2 At the **Label** text field, type Down Nose.
3 Locate the **Control Points** section. In the text fields, enter the following settings:

	xw	yw	
1:	0	0	m
2:	a-0.005	MaxTh/2	m
3:	a	MaxTh	m

4 Locate the **Weights** section. At the **2:** text field type 1.

Up Nose (qb2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** right-click **Down Nose (qb1)** and choose **Duplicate**.
2 At the **Label** text field, type Up Nose.
3 Locate the **Control Points** section. In the text fields, enter the following settings:

	xw	yw	
1:	0	0	m
2:	-(a-0.005)	MaxTh/2	m
3:	-a	MaxTh	m

Down Tail (qb3)

1 In the **Model Builder** window, under Com**ponent 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** right-click **Up Nose (qb2)** and choose **Duplicate**.
2 At the **Label** text field, type Down Tail.
3 Locate the **Control Points** section. In the text fields, enter the following settings:

	xw	yw	
1:	a	MaxTh	m
2:	a-(Width-MaxTh)/40	MaxTh+(Width-MaxTh)/2	m
3:	0	Width	m

Up Tail (qb4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** right-click **Up Nose (qb2)** and choose **Duplicate**.
2 At the **Label** text field, type Down Tail.
3 Locate the **Control Points** section. In the text fields, enter the following settings:

	xw	yw	
1:	-a	MaxTh	m
2:	-(a-(Width-MaxTh)/40)	MaxTh+(Width-MaxTh)/2	m
3:	0	Width	m

Convert to Solid (csol1)

1 In the **Work Plane** toolbar, click **Conversions** and choose **Convert to Solid**.
2 Locate the **Input** section. Click **Paste Selection**.
3 In the **Paste Selection** dialog box, type qb1, qb2, qb3, qb4.
4 Click **OK**.
5 At the **Setting** window for **Convert to Solid**, click **Build All**.

Fillet Sides (fil1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Fillet**.
2 At the **Label** text field, type Fillet Sides.
2 Locate the **Points** section. Click **Paste Selection**.
3 In the **Paste Selection** dialog box, type csol1 1, csol1 4.
4 Click **OK**.
5 Locate the **Radius** section. At the **Radius** text field, type 0.1.

Fillet Tip (fil2)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Fillet**.
2 At the **Label** text field, type Fillet Tip.
3 Locate the **Points** section. Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type fil1 3.
5 Click **OK**.
6 Locate the **Radius** section. At the **Radius** text field, type 0.001*Cu.

Extrude 1 (ext1)

1 In the **Geometry** toolbar, click **Extrude**.
2 Locate the **Distances** section. Under the **Distances (m)** column, type Length.

Rotate 1 (rot1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Rotate**.
2 Locate the **Input** section and click **Paste Selection**.
3 In the **Paste Selection** dialog box, type ext1.
4 Click **OK**.
5 Locate the **Rotation** section. At the **Angle** text field, type -90+Angle.

Difference 1 (dif1)

3 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
4 Locate the **Objects to add** section and click **Paste Selection**.
5 In the **Paste Selection** dialog box, type blk1.
6 Click **OK**.
7 Locate the **Objects to subtract** section. Click **Paste Selection**.
8 In the **Paste Selection** dialog box, type rot1.
9 Click **OK**.
10 At the top of the **Settings** window click **Build All Objects**.

Laminar Flow (spf)

Inlet Air

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.
2 At the **Label** text field type Inlet Air.
3 Locate the **Boundary Selection** section and click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 1.
5 Click **OK**.
6 Locate the **Velocity** section and at the text field for $\mathbf{U_0}$ type V_Air.

Outlet

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Outlet**.
2 Locate the **Boundary Selection** section and click **Paste Selection**.
3 In the **Paste Selection** dialog box, type 15.

Materials

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Air**.
4 Click **Add to Component** in the window toolbar.
5 Close the **Add Material** window.

Study 1

Parametric Sweep

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Study 1** and choose **Parametric Sweep**.
2 In the **Settings** window for **Parametric sweep**, locate the **Study Setting** section.
3 Click **Add**.
4 Under the **Parameter** column, from the list choose **Angle (Angle of attack)**.
5 Click **Range**.
6 In the **Range** window.
7 At the **Start** text field, type 0.
8 At the **Step** text field, type 5.
9 At the **Stop** text field, type 35.
10 Click **Add**.
11 Under the **Parameter unit**, type deg.

Job Configurations

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Study 1** and choose **Show More Options…**.
2 Select the **Study>Solver and Job Configurations** check box.
3 Click OK.
4 In the **Model Builder** window, under **Component 1 (comp1)>Study 1** right-click **Job Configurations** and choose **Parametric Sweep**.
5 Locate the **General** section. From the **Defined by study step** list, choose **Parametric Sweep.**
6 In the **Model Builder** window, under **Component 1 (comp1)>Study 1>Job Configuration** right-click **Parametric Sweep 1** and choose **Enable**.
7 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Study 1** and choose **Get Initial Value**.
8 In the **Model Builder** window, under **Component 1 (comp1)>Study 1>Job Configuration** click **Parametric Sweep 2.**
9 Locate the **Error** section. Unselect the **Stop if error** check box.

Results

Surface Integration

1 In the **Model Builder** window, under **Component 1 (comp1)>Results** right-click **Derived Values** and at the **Integration** option, choose **Surface Integration**.
2 Locate the **Selection** section. Click **Paste Selection**.
3 At the **Paste Selection** text dialog box, type 6-14.
4 Click **OK**.
5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Descripltion
-spf.T_stressx*2/(rho_Air*V_Air^2)	m^2	Drag Coeff*CS
spf.T_stressy*2/(rho_Air*V_Air^2)	m^2	Lift Coeff*CS
-nx*(nx<0)	m^2	Cross Section (CS)

Compute

1 In the **Model Builder** window, under **Component 1 (comp1)>Study 1** click **Parametric Sweep**.
2 Click **Compute**.

Evaluate results

1 In the **Error** window click OK.
2 In the **Model Builder** window, under **Component 1 (comp1)>Results>Derived Values** click **Surface Integration 1**.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**
4 At the top of the **Settings** window, click **Evaluate**.

In order to plot the results, you can use Origin Lab or Excel using as x-Axis the **Angle (deg)** values and y-Axis the **Drag Coeff*CS (m^2)** and **Lift Coeff*CS (m^2)** values multiplied with **CS (m^2)** values.

A computed version of this model can be found on:
dropbox.com/scl/fo/361lf7jl80prwqn42l997/h?dl=0&rlkey=abx80noagehwnzt5nw7mhnm7n

6.3 Results

The Lift and drag Coefficients curves are shown at the following Figure 6. 2.

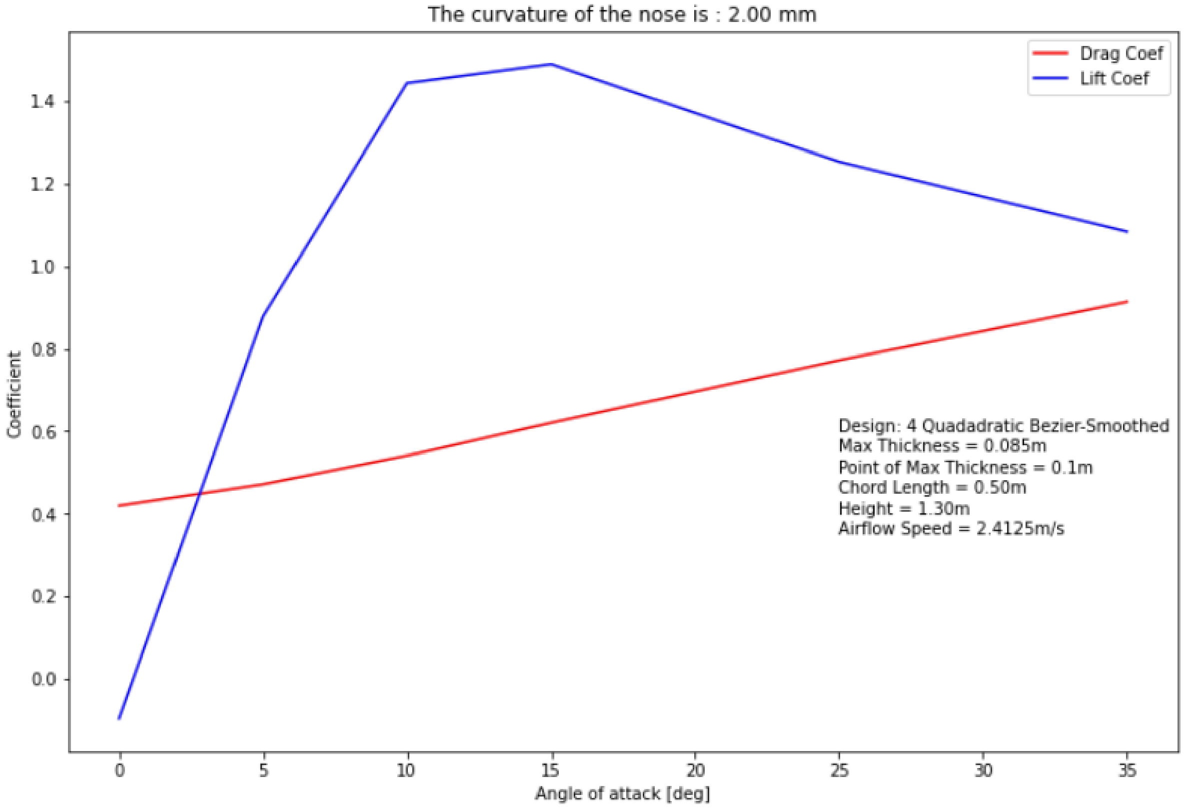


Figure 6. 2

6.4 Use in real-time sailboat

This design was used for the construction of the sailboat shown below at the Figure 6. 3.

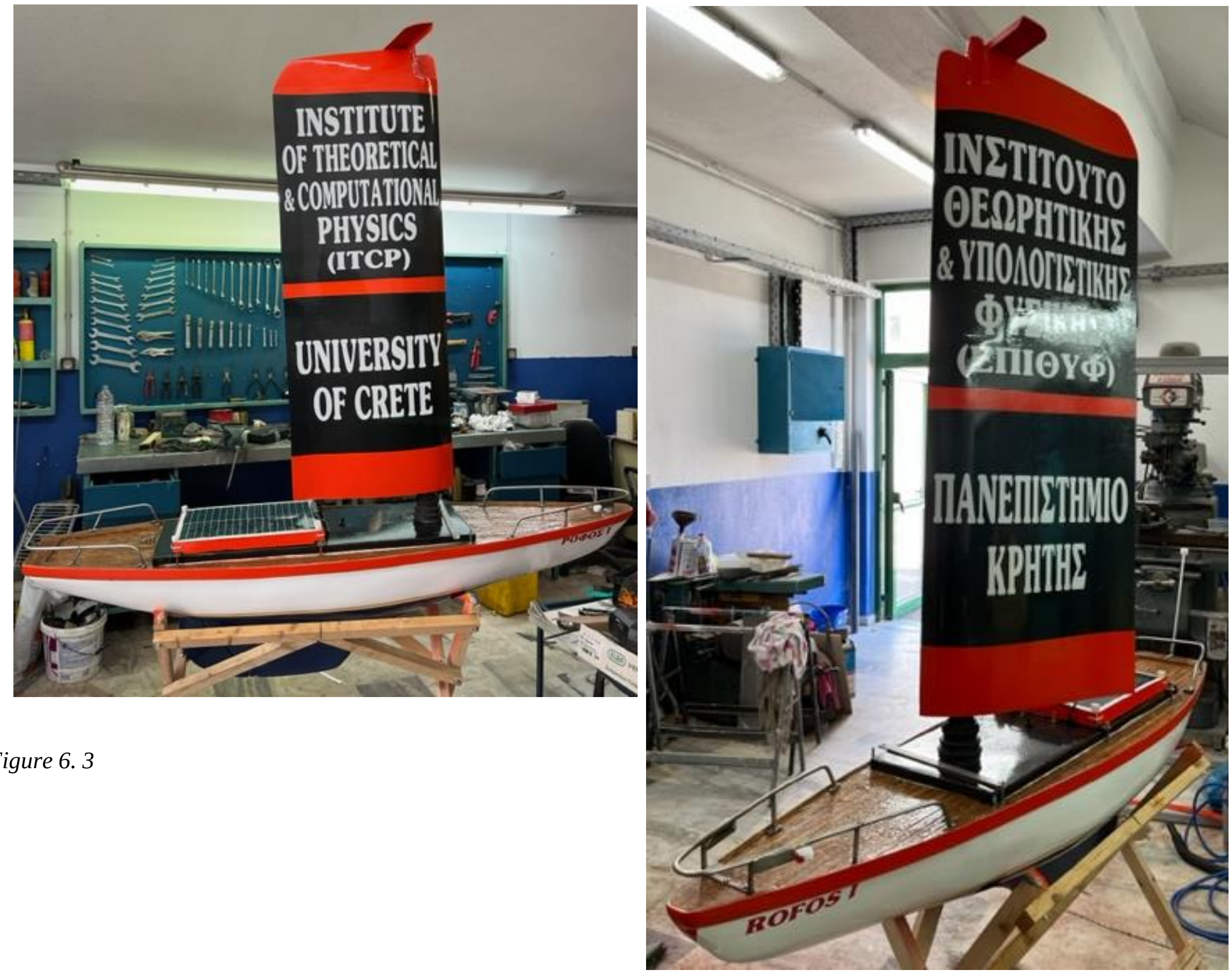


Figure 6. 3

7 Sailboat in Wind-Water flow environment

7.1 Introduction

This is a coupled Water-Wind environment with a material sailboat sailing. It is a model with time evolution of the flow of air and water interfering with a free moving material sailboat. The model makes the transition from stationary system to full speed flow (5m/s airflow speed, 1.25m/s water flow speed) but after a while it crashes before it can reach equilibrium. A screenshot of a timestep can be seen at the Figure 7.1. 1 below.

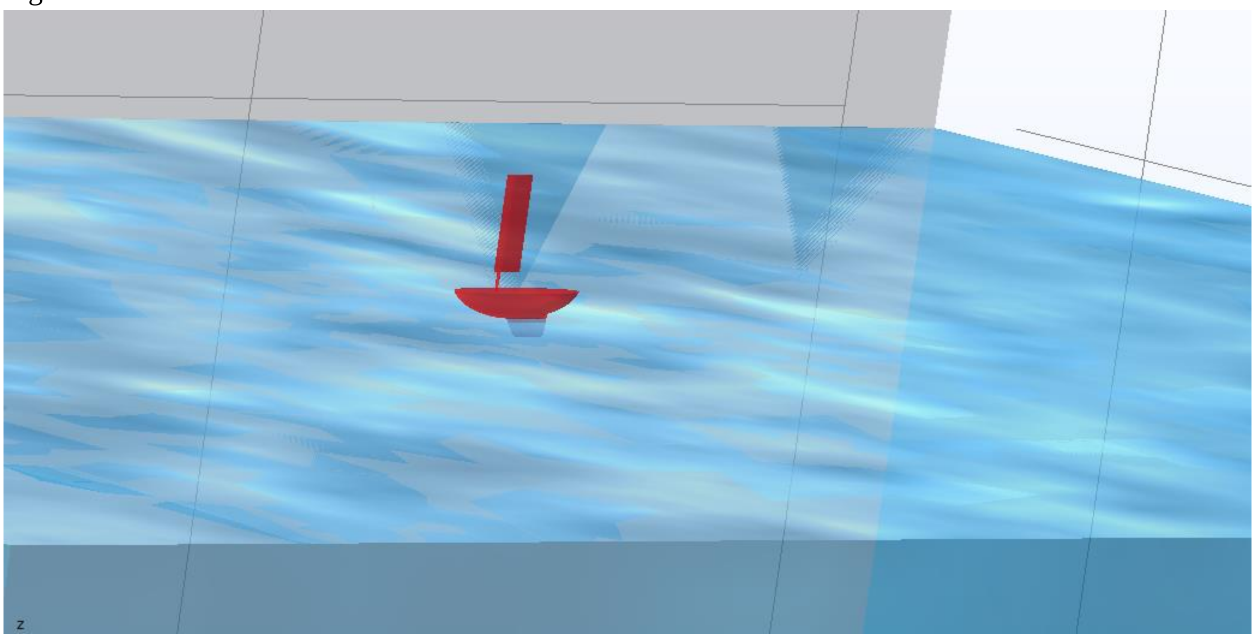


Figure 7.1. 1

The best design from the previous investigation was used together with a basic design for the rest of the sailboat. The model has been structured in order the sailboat's sail to rotate in relation to the timestep or the tilt of the sailboat or any other variable that could be introduced.
Other than the Laminar flow interface that we had before, we are using also "Moving Mesh", "Multibody Dynamics" and "Phase Field" modules, together with the "Two-Phase Flow, Phase Field", "Fluid-Structure Interaction, Pair" Multiphysics modules.

- The **Moving Mesh** features controls the spatial frame and we are using them to model the deformations where the geometry changes its shape due to motion of solid boundaries and deformation of solid domains [17].
- The **Multibody Dynamics** interface is used to model mechanical assemblies [18]. With this interface we model the solid Sailboat, track its deformation in space and the rotation of the sail in respect to the sailboat.
- The **Phase Field** interface is used to track moving interfaces by solving two transport equations, one for the phase field variable, φ, and one for the mixing energy density, ψ. The position of the interface is determined by minimizing the free energy [19]. With this interface we define the moving boundary between the two fluids, the water and the air.

- The **Two-Phase Flow, Phase Field** multiphysics coupling feature defines the density and dynamic viscosity of the fluid used in the Laminar Flow and Turbulent Flow interfaces and it defines the surface tension on the interface in form of a volume force used in the momentum equation. It also enables the Phase Field interface to use the velocity field calculated from the Laminar Flow or Turbulent Flow interface to transport the interface [20].
- The **Fluid-Structure Interaction, Pair** multiphysics node provides a coupling between a pair of boundaries, where one is the boundary of a fluid domain and the other is part of solid structure [21].

This model was inspired by the “Mechanism Submerged in Fluid” model [22]. That model uses the “Fluid-Structure Interaction, Pair” Multiphysics interface with a flow of one fluid instead of two the model described below uses.
The model was built in COMSOL Multiphysics 6.0.

7.2 Model Instructions

The model was built with the instructions presented below:

New
In the **New** window, click **Model Wizard**.

Model Wizard

1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Fluid Flow>Multiphase Flow>Two-Phase Flow, Phase Field>Laminar Flow**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Fluid Flow> Fluid-Structure Interaction>Fluid-Multibody Interaction, Assembly**.
5. Click **Add**.
6. Locate the **Added physics interfaces** section. Select **Laminar Flow (spf2)** and click **Remove**.
7. Click **Study**.
8. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Time Dependent with Phase Initialization**.
9. Click **Done**.

Global Definitions

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
V_Max	5[m/s]	5 m/s	Max flow speed
th_max	15[deg]	0.2618 rad	Maximum Sail rotation

Geometry 1

1 In the Graphics window, click the **Transparency** button.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 Click **Browse**.
4 Browse and double click the file Best_Sail_Sailboat.mphbin.
5 Click **Import**.

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.
2 In the **Settings** window for **Copy**, locate the **Input Objects** section. Click **Paste Selection**.
3 In the **Paste Selection** dialog box, type imp1(1) imp1(2).
4 Click **OK**.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type 200.
4 In the **Depth** text field, type 120.
5 In the **Height** text field, type 160.
6 Locate the **Position** section. From the **Base** list, choose **Center**.

Work Plane 1 and 2

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Geometry** toolbar, click again **Work Plane**.
3 Locate the **Plane Definition** section. From the **Plane** list choose **zx- plane**.

Partition Domains 1

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.
2 Locate the **Partition Faces** section. Under the **Faces to Partition** subsection, click **Paste Selection**.
3 In the **Paste Selection** dialog box, type blk1 1.
4 Click **OK**.
5 At the **Work plane** list, choose **Work Plane 1 (wp1)**.
6 At the **Setting** window for **Partition Domains**, click the **Build Selected** button.

Partition Domains 1

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.
2 Locate the **Partition Faces** section. Under the **Faces to Partition** subsection, click **Paste Selection**.
3 In the **Paste Selection** dialog box, type pard1 1, pard1 2.
4 Click **OK**.
5 At the **Setting** window for **Partition Domains**, click the **Build Selected** button.

Work Plane 2

1. In the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3. At the **z-coordinate** text field, type 30.
4. In the **Geometry** toolbar, click again **Work Plane**.
5. In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
6. At the **z-coordinate** text field, type -30.

Partition Faces 1

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Faces**.
2. In the **Settings** window for **Partition Faces**, locate the **Partition Faces** section.
3. Under the **Faces to Partition subsection**, click **Paste Selection**.
4. In the **Paste Selection** dialog box, type pard2 18, pard2 20.
5. Click **OK**.
6. At the **Partition with** list, choose **Work Plane**.
7. At the **Work plane** list, choose **Work Plane 3 (wp3)**.

Partition Faces 2

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Faces**.
2. In the **Settings** window for **Partition Faces**, locate the **Partition Faces** section.
3. Under the **Faces to Partition subsection**, click **Paste Selection**.
4. In the **Paste Selection** dialog box, type parf1 17, parf1 20.
5. Click **OK**.
6. At the **Partition with** list, choose **Work Plane**.
7. In the **Partition Faces** window, click **Build Selected**.

Block II (blk2)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type 20.
4. In the **Depth** text field, type 120.
5. In the **Height** text field, type 1.
6. In the **Settings** window for **Block**, locate the **Position** section.
7. From the **Base** list, choose **Center**.
8. In the **x** text field, type -90.
9. In the **y** text field, type 0.
10. In the **z** text field, type 0.5.

Difference 1 (dif1)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2. Locate the **Objects to add** section and click **Paste Selection**.
3. In the **Paste Selection** dialog box, type parf2.
4. Click **OK**.
5. Locate the **Objects to subtract** section. Click **Paste Selection**.
6. In the **Paste Selection** dialog box, type copy1(1), copy1(2), blk2.
7. Click **OK**.
8. In the **Difference** window, click **Build Selected**.

Form Union (fin)

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry I** click **Form Union (fin)**.
2. In the **Settings** window for **Form Union/Assembly**, locate the Form **Union/Assembly** section.
3. From the **Action** list, choose **Form an assembly**.
4. In the **Settings** window for **Form Union/Assembly**, click **Build All**.

Definitions

Identity Boundary Pair 1 (ap1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Definitions,** click **Identity Boundary Pair 1 (ap1)**.
2. In the **Settings** window for **Pair**, locate the **Source Boundaries** section.
3. Click **Create Selection**.
4. In the **Create Selection** dialog box, type Fluid Boundaries (Boat).
5. Click **OK**.
6. In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.
7. Click **Create Selection**.
8. In the **Create Selection** dialog box, type Solid Boundaries (Boat).
9. Click **OK**.
10. In the **Settings** window for **Pair**, locate the **Frame** section.
11. From the **Source frame** list, choose **Material (X, Y, Z)**.
12. From the **Destination frame** list, choose **Material (X, Y, Z)**.

Identity Boundary Pair 2 (ap2)

1. In the **Model Builder** window, under **Component 1 (comp1)>Definitions,** click **Identity Boundary Pair 2 (ap2)**.
2. In the **Settings** window for **Pair**, locate the **Source Boundaries** section.
3. Click **Create Selection**.
4. In the **Create Selection** dialog box, type Fluid Boundaries (Sail).
5. Click **OK**.
6. In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.
7. Click **Create Selection**.
8. In the **Create Selection** dialog box, type Solid Boundaries (Sail).
9. Click **OK**.
10. In the **Settings** window for **Pair**, locate the **Frame** section.
11. From the **Source frame** list, choose **Material (X, Y, Z)**.
12. From the **Destination frame** list, choose **Material (X, Y, Z)**.

All Fluid Boundaries

1. In the **Definitions** toolbar, click **Union**.
2. In the **Settings** window for **Union**, type All Fluid Boundaries in the **Label** text field.
3. Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4. Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
5. In the **Add** dialog box, in the **Selections to add** list, choose Fluid Boundaries (Boat) and Fluid Boundaries (Sail).
6. Click **OK**.

All Solid Boundaries

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type All Solid Boundaries in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
5 In the **Add** dialog box, in the **Selections to add** list, choose Solid Boundaries (Boat) and Solid Boundaries (Sail).
6 Click **OK**.

Step I (Step1)

1 In the **Home** toolbar, click **Functions** and choose **Local>Step**.
2 Locate the **Parameters** section. In the **Location** text field, type 0.05.

Analytic 1 (an1)

1 In the **Home** toolbar, click **Functions** and choose **Local>Analytic**.
2 In the **Settings** window for **Analytic,** locate the **Definition** section.
3 In the **Expression** text field, type th_max*(sin(2*pi*1*t)*(t<0.25)*step1(t)+(t>=0.25)).
4 In the **Arguments** text field, type t.
5 Locate the **Units** section.
6 In the **Function** text field, type rad.
7 In the table, enter the following settings:

Arguments	Unit
t	s

Analytic 2 (an2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions,** right-click **Analytic 1 (an1)** and choose **Duplicate**.
2 In the **Settings** window for **Analytic**, locate the **Definition** section.
3 In the **Expression** text field, type V_Max*((t<0.25)*step1(t)+(t>=0.25)).
4 Locate the **Units** section. In the **Function** text field, type m/s.

Variables 1

1 In the **Model Builder** window, under Component 1 (comp1) right-click **Definitions** and choose **Variables**.
2 In the Settings window for **Variables**, locate the **Variables** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
th	an1(t)	rad	Sail rotation
V_Inlet	an2(t)	m/s	Flow speed

Laminar Flow (spf)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.
3 Select 5 and 6 and click **Remove from Selection**.
4 Locate the **Physical Model** section, under **Compressibility**. Select the **Include gravity** check box.

Wall 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Wall 1**.
2. In the **Settings** window for **Wall**, locate the **Boundary Condition** section.
3. From the **Wall condition** list, choose **Slip**.

Pressure Point Constraint 1

1. In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2. In the **Settings** window for **Pressure Point Constrain**, locate the **Point Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog box, type 1.
5. Click **OK**.

Inlet Air

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, at the **Label** text field type Inlet Air.
3. Locate the **Boundary Selection** section and click **Paste Selection**.
4. In the **Paste Selection** dialog box, type 5, 13.
5. Click **OK**.
6. Locate the **Velocity** section and at the text field for $\mathbf{U_0}$ type V_Inlet.

Inlet Water

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, at the **Label** text field type Inlet Water.
3. Locate the **Boundary Selection** section and click **Paste Selection**.
4. In the **Paste Selection** dialog box, type 1, 9.
5. Click **OK**.
6. Locate the **Velocity** section and at the text field for $\mathbf{U_0}$ type V_Inlet/4.

Outlet

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Outlet**.
2. n the **Settings** window for **Outlet**, locate the **Boundary Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog box, type 69, 72, 73, 76.

Phase Field (pf)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Phase Field (pf)**.
2. In the **Settings** window for **Phase Field**, locate the **Domain Selection** section. Select 5 and 6.
3. Click **Remove from Selection**.

Initial Values, Fluid 2

1. In the **Model Builder** window, under **Component 1 (comp1)>Phase Field (pf)** click **Initial Values, Fluid 2**.
2. Locate the **Domain Selection** section. Click **Paste Selection**.
3. In the **Paste Selection** dialog box, type 1, 3.

Inlet Air

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Phase Field (pf)** and choose **Inlet**.
2 In the **Settings** window for **Inlet**, at the **Label** text field type Inlet Air.
3 Locate the **Boundary Selection** section and click **Paste Selection**.
4 In the **Paste Selection** dialog box, type 5,13.
5 Click **OK**.

Inlet Water

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Phase Field (pf)** and choose **Inlet**.
2 In the **Settings** window for **Inlet**, at the **Label** text field type Inlet Water.
3 Locate the **Boundary Selection** section and click **Paste Selection**.
4 In the **Paste Selection** dialog box, type 1, 9.
5 Click **OK**.
6 Locate the **Phase Field Condition** section and at the list choose **Fluid 2 (φ = 1)**.

Outlet

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Deforming Domain 2** and choose **Delete**.
3 Locate the **Boundary Selection** section and click **Paste Selection**.
4 In the **Paste Selection** dialog box, type 69, 72, 73, 76.

Multibody Dynamics (mbd)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Multibody Dynamics (mbd)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.
3 Select 1, 2, 3, 4 and click **Remove from Selection**.

Gravity 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multibody Dynamics (mbd)** and from the **Volume Forces** list, choose **Gravity**.
2 In the **Settings** window for **Gravity**, locate the **Domain Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type 5, 6.
5 Click **OK**.

Sailboat

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multibody Dynamics (mbd)** and from the **Material Model** list, choose **Rigid Domain**.
2 In the **Settings** window for Rigid Domain, at the **Label** text field type Sailboat.
3 Locate the **Domain Selection** section and click **Paste Selection**.
4 In the **Paste Selection** dialog box, type 5, 6.
5 Click **OK**.

Materials

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Air**.
4 Click **Add to Component** in the window toolbar.
5 In the tree, select **Built-in>Water, liquid**.
6 Click **Add to Component** in the window toolbar.
7 In the tree, select **Built-in>Structural steel**.
8 Click **Add to Component** in the window toolbar.
9 Close the **Add Material** window.

Water, liquid (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.
2 Locate the **Geometric Entity Selection** section and click **Paste Selection**.
3 In the **Paste Selection** dialog box, type 1, 3.
4 Click **OK**.

Structural steel (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Structural steel (mat3)**.
2 Locate the **Geometric Entity Selection** section and click **Paste Selection**.
3 In the **Paste Selection** dialog box, type 5, 6.
4 Click **OK**.
5 Locate the **Material Contents** section.
6 At the **Density** row, under the **Value** column type 1050[kg/m^3].

Moving Mesh

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Moving Mesh** click **Deforming Domain 1**.
2 In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.
3 Click **Paste selection**.
4 In the **Paste Selection** dialog box, type 1-4.
5 Click **OK**.

Prescribed Mesh Displacement 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Moving Mesh** and choose **Prescribed Mesh Displacement**.
2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All Fluid Boundaries**.
4 Locate the **Prescribed Mesh Displacement** section. Specify the **dx** vector as:

fsip1.u_solid	**X**
fsip1.v_solid	**Y**
fsip1.w_solid	**Z**

Multiphysics

Two-Phase Flow, Phase Field 1 (tpf1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Two-Phase Flow, Phase Field 1 (tpf1)**.
2. In the **Settings** window for **Two-Phase Flow, Phase Field**.
3. Locate the **Fluid 1 Properties** section. From the **Fluid 1** list, choose **Air (mat1)**.
4. Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, choose **Water, liquid (mat2)**.

Fluid-Structure Interaction, Pair 1 (fsip1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Fluid-Structure Interaction, Pair 1 (fsip1)**.
2. In the **Settings** window for **Fluid-Structure Interaction**, locate the **Fixed Geometry** section. From the **Fixed geometry coupling type** list, choose **Fluid loading on structure.**
3. Locate the **Pair Selection** section. Under **Pair**, click **Add**.
4. In the **Add** dialog box, choose **Identity Boundary Pair 1 (ap1)** and **Identity Boundary Pair 2 (ap2)**.
5. Click **OK**.

Mesh 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Sequence** section and choose **User-controlled mesh**.

Size 2

1. In the **Model Builder** window, under **Component 1 (comp1)**>**Mesh 1** right-click **Size 1** and choose **Duplicate**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section. From the **Geometric entity level** list choose **Boundary**.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **All Solid Boundaries**.
4. Locate the **Element Size** section. From the **Predefined** list, choose **Normal**.

Size 1

1. In the **Model Builder** window, under **Component 1 (comp1)**>**Mesh 1** and click **Size 1**.
2. Locate the **Element Size** section. From the **Predefined** list, choose **Coarse**.

Corner Refinement 1

1. In the **Model Builder** window, under **Component 1 (comp1)**>**Mesh 1** and click **Corner Refinement 1**.
2. Locate the **Boundary Selection** section and click **Clear Selection**.

Boundary Layer Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** Click to expand **Boundary Layers 1**.
2 Click **Boundary Layer Properties 1**.
3 In the Settings window for Boundary Layers Properties, locate the **Boundary Selection** section.
4 From the **Selection** list, choose **All Solid Boundaries**.
5 Locate the **Layers** section.
6 At the **Number of layers:** text field, type 8.

Study 1

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.
2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3 In the **Output times** text field, type range(0,0.01,1.5).

Default Solver

1 In the **Model Builder** window, under **Study 1** right-click **Solver Configurations** and choose **Show Default Solver**.

Fully Coupled

2 In the **Model Builder** window, under **Study 1<Solver Configurations** click to expand **Solution 1 (sol1)**.
3 Click to expand **Stationary Solver 1** and click at **Fully Coupled 1**.
4 At the **Settings** window for **Fully Coupled**, locate the **General** section.
5 From the **Linear solver:** list choose **Direct, interface distance (pf)**.

Phase field variables

1 In the **Model Builder** window, under **Study 1<Solver Configurations** click to expand **Time-Dependent Solver 1**.
2 Click to expand **Segregated 1** and click **Phase field variables**.
3 In the Settings for **Segregated Step**, locate the **General** section.
4 From the **Linear solver:** list, choose **Direct, phase field variables (pf)**.

Compute

1 In the **Model Builder** window click **Study 1**.
2 In the **Setting** window for Study, click **Compute**

After Computation, at the **Error** window, click **OK**.

Results

Volume Fraction of Water

1 In the **Model Builder** window right-click **Results** and choose **3D Plot Group**.
2 At the **Label** text field, type Volume Fraction of Water.
3 Locate the **Plot Setting** section. Unselect the **Plot Dataset edges** check box.

Water Appearance

1. In the **Model Builder** window under **Results** right-click **Volume Fraction of Water** and choose **Isosurface**.
2. At the **Label** test field, type Water Appearance.
3. Locate the **Expression** section. At the **Expression** text field, type pf.Vf2.
4. Locate the **Levels** section.
5. At the **Entry method** list, choose **Levels**.
6. At the **Levels** text field type 0.5.
7. Locate the **Coloring and Style** section. At the **Isosurface type** list choose **Filled**.
8. In the **Model Builder** window under **Results>Volume Fraction of Water** right-click **Water Appearance** and choose **Material Appearance**.

Sailboat Appearance

1. In the **Model Builder** window under **Results** right-click **Volume Fraction of Water** and choose **Surface**.
2. At the **Label** test field, type Sailboat Appearance.
3. Locate the **Expression** section. At the **Expression** text field, type mbd.disp.
4. Locate the **Coloring and Style** section.
5. At the **Coloring** list choose **Uniform**.

Wall Appearance

1. In the **Model Builder** window under **Results** right-click **Volume Fraction of Water** and choose **Surface**.
2. At the **Label** test field, type Wall Appearance.
3. Locate the **Expression** section. At the **Expression** text field, type 1.
4. Locate the **Coloring and Style** section.
5. At the **Coloring** list choose **Uniform**.
6. At the **Color** list, choose **Grey**.
7. In the **Model Builder** window under **Results>Volume Fraction of Water** right-click **Wall Appearance** and choose **Selection**.
8. Locate the **Selection** section and click **Paste Selection**.
9. Type 3, 11, 17, 18 and click **OK**.
10. At the top of the **Settings** window for **Selection**, click **Plot**.

Sailing Animation

1. In the **Model Builder** window under **Results** click **Volume Fraction of Fluid 2**.
2. In the **Results** toolbar, click **Animation** and choose **Player.**
3. At the **Label** test field, type Sailing Animation.
4. Click the **Play** button in the **Graphics** toolbar.

A computed version of this model can be found on:
dropbox.com/scl/fo/361lf7jl80prwqn42l997/h?dl=0&rlkey=abx80noagehwnzt5nw7mhnm7n

7.3 Commentary

- The “Two-Phase Flow, Phase Field” multiphysics interface was tested before adding the “Fluid-Structure Interaction, Pair” multiphysics interface with different configurations for simultaneous flow of two fluids and it reaches equilibrium. Creating a model that reaches equilibrium with the full set of interfaces would require investigation on the solid Sailboat and the interference between the multiphysics interfaces.
- The outlet surface is smaller than the Inlets in order to restrict the outflow and create a positive pressure. The positive pressure makes the movement of the particles of the fluids less arbitrary and that helps with the computation.
- The inlets of the fluids have to be separated otherwise instabilities are created. This is true also for the outlets.
- The block that was added helps to avoid the great instabilities that are being created at the barrier between the inlets.
- The speeds of the Air and Water flows was investigated and selected as the ones that help the model crash as late as possible.
- A step function has been introduced in order to smoothen the initiation of the inflow and avoid the instabilities that follows from an instantaneous inflow.
- The density of the material used for the Sailboat was reduced to match the overall weight of the real-time sailboat, as it will be mostly hollow, and to simplify the model by reducing the movement of the sailboat from gravity.
- The “Fluid-Structure Interaction, Pair” multiphysics interface was simplified by computing only the fluid load on structure as that is what we are interested in.
- The mesh was configured in order to reduce the unnecessary mesh refinement to the regions that we are not intended to take measurements. This makes the computation a lot faster.

8 Future Steps

- Configure the “Sailboat in Wind-Water flow environment” in order to reach equilibrium by investigating the solid Sailboat and the interference between the multiphysics interfaces.
- Test the sail designs in the “Sailboat in Wind-Water flow environment” in respect to the maximum speed the sailboat can reach.
- Optimize the rest of the sailboat in respect to the maximum speed the sailboat can reach.
- Configure the sail to rotate in relation to the sailboat’s positioning and tilt and to the environment’s conditions in order to navigate to a destination.

9 References

[1] Comsol Multiphysics, „Navier-Stokes Equations,” [Online]. Available: https://www.comsol.com/multiphysics/navier-stokes-equations.

[2] Cadence CFD Solutions, „The Dimensionless Navier-Stokes Equation,” [Online]. Available: https://resources.system-analysis.cadence.com/blog/msa2022-the-dimensionless-navier-stokes-equation.

[3] Wikipedia, „Reynolds number,” [Online]. Available: https://en.wikipedia.org/wiki/Reynolds_number.

[4] G. Falkovich, in *Fluid Mechanics*, Cambridge University Press, 2018.

[5] N. Hall, in *Boundary Layer*, Glenn Research Center, 2015.

[6] Comsol multiphysics, „The Laminar Flow Interface,” [Online]. Available: https://doc.comsol.com/5.5/doc/com.comsol.help.cfd/cfd_ug_fluidflow_single.06.008.html.

[7] A. Sommerfeld, in *A Contribution to Hydrodynamic Explanation of Turbulent Fluid Motions*, 1908, pp. 116-124.

[8] B. W. McCormick, „Aerodynamics, Aeronautics, and Flight Mechanics.,” 1979.

[9] Wikipedia, „Drag coefficient,” [Online]. Available: https://en.wikipedia.org/wiki/Drag_coefficient#cite_note-3.

[10] L. J. Clancy, „Aerodynamics,” 1975.

[11] I. H. a. D. A. E. Abbott, „Theory of Wing Sections.”.

[12] L. J. Clancy, Aerodynamics.

[13] NASA, „Lift to Drag Ratio,” [Online]. Available: https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/lift-to-drag-ratio/.

[14] F. A. Morrison, „Data Correlation for Drag Coefficientfor Sphere,” 2016. [Online]. Available: https://pages.mtu.edu/~fmorriso/DataCorrelationForSphereDrag2016.pdf.

[15] T. Baracu, „Computational analysis of the flow around a cylinder and of the drag force,” 2011. [Online]. Available: https://www.researchgate.net/publication/274309261_Computational_analysis_of_the_flow_around_a_cylinder_and_of_the_drag_force.

[16] S. H. a. T. R. Sudarsono*, „Prediction of Aerodynamics Coefficients of Modified NACA 4415 Airfoil Using Computational Fluid Dynamics,” 2020. [Online]. Available: https://www.e3s-conferences.org/articles/e3sconf/pdf/2020/62/e3sconf_icenis2020_11002.pdf.

[17] Comsol Multiphysics, „Moving Mesh Features,” [Online]. Available: https://doc.comsol.com/5.5/doc/com.comsol.help.comsol/comsol_ref_deformedmeshes.25.14.html.

[18] COmsol Multiphysics, „The Multibody Dynamics Interface,” [Online]. Available: https://doc.comsol.com/5.5/doc/com.comsol.help.mbd/mbd_ug_multibody.5.02.html.

[19] Comsol Multiphysics, „The Phase Field Interface,” [Online]. Available:

https://doc.comsol.com/6.0/doc/com.comsol.help.cfd/cfd_ug_math_levelset_phasefield.14.11.html.

[20] Comsol Multiphysics, „The Two-Phase Flow, Phase Field Coupling Feature,” [Online]. Available: https://doc.comsol.com/6.0/doc/com.comsol.help.cfd/cfd_ug_fluidflow_multi.09.015.html#2325052.

[21] Comsol Multiphysics, „Fluid-Structure Interaction, Pair,” [Online]. Available: https://doc.comsol.com/6.0/doc/com.comsol.help.sme/sme_ug_multiphysics.15.40.html.

[22] Comsol Multiphysics, [Online]. Available: https://www.comsol.com/model/mechanism-submerged-in-fluid-70251.