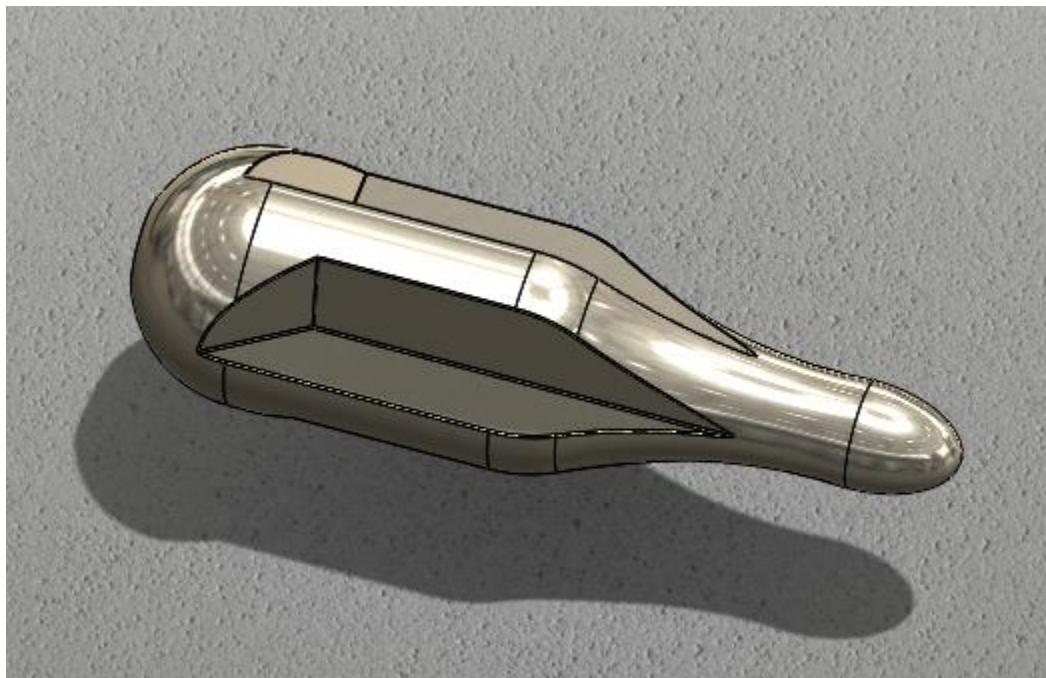


Rotatory MiniPat



By: Diego Ricardo Higuera Ruiz
Instructor's Name: Michael Shafer



Department of Mechanical Engineering
Northern Arizona University
Flagstaff, AZ 86011

ACKNOWLEDGMENTS

This thesis becomes a reality with the kind support and help of many individuals. I would like to extend my sincere thanks to all of them.

First of all, I would like to express the deepest appreciation to my family (Maria José, Diego Estanislado and Maria) and friends (Manuel Jesús, José Manuel and David), without whom, this thesis would not have been written due to their unconditional support.

To University of Jaén and Northern Arizona University, which have provided economical and administrative support.

From a closer point of view to this project, I would like to be thankful to my supervisor Dr. Michael Shafer, who has professionally guided in this project.

To Dr. Thomas Acker, who was my professor of CFD and Aerodynamics, thank you so much for being such a positive person and make me enjoy these classes, which have been very fruitful and useful to my thesis.

And last, but not least, to Gregory Hahn and Seth Lawrence, who have been totally into this project and spent their precious time.

Abstract.

The studies of migratory movement of some marine species is currently being an important subject, these allow us to discover study movements, habitat utilization and post -release survival of pelagic animals. With the aim to achieve this, a sensor called GPS is placed on marine animals. The mission of this sensor is to inform the global position of the animal anywhere it is.

To accomplish this goal, many firms in this market have presented different systems and products. WILDLIFE COMPUTERS is a powerful company, which designs and develops these GPSs. For our case, let us focus on the MiniPAT- 348A. This device features low-drag shape and pinger for radio tracking recovery. Sensor data are collected during deployment, and archived in onboard memory, finally the final data is uploaded to Argos satellites. Nevertheless, the main issue of this sensor is its operating life time, which is about 2 years.

Therefore, our mission in this project is to improve the operating life, reaching the self-supply of the whole system and avoiding replacing batteries

In order to keep the battery charged, it needs to be developed an energy source. Here is where our project takes place.

This project consists in a rotatory device, which is rotated by water going through. The rotational motion produce energy necessary to power the whole system. Our goals are to calculate the main parameters which characterize this device by various means: Simulations, theoretic and experimental methods.

The final part of this project would be to install an electric motor (generator) inside of the device as well as the battery and sensor. Finally we will get the whole system in one, however it is important to mention the fact of having different ways of assembling the whole system.

Index

Table of Contents

Introduction	4
Objectives.....	5
Fluid Mechanics Fundaments.....	5
Physical Interpretation.....	5
Propellers.....	9
Dimensional Analysis.....	14
Methods	16
Design of a rotatory device	16
Simulation. Ansys (Fluent)	20
Two Dimensional simulation.....	20
Three Dimensional simulation. Non-Fluted	27
Three Dimensional simulation. Fluted	44
Experiments.....	54
Density, wind tunnel velocity and force-meter calibration	54
Dimensional Analysis and Flow similarity	57
Rapid Prototype	61
Theoretical torque calculation.....	64
Results & Discussions	66
Simulation. Ansys (Fluent)	66
Two Dimensional simulation.....	66
Three Dimensional simulation. Non-Fluted	68
Three Dimensional simulation. Fluted	74
Experimental Results	81
Conclusions	84
References	85
Appendix	86

Introduction.

• Objectives.

As we have briefly talked in the abstract, the motivation to develop this project is to find an optimal way to create tracking sensors with an operating life forever. Therefore, the main idea is to design a rotated device and produce energy due to its angular speed and torque. Also we have to take into account drag forces and streamline body.

Below, it can be seen at illustration [1], how the device is going to look like:

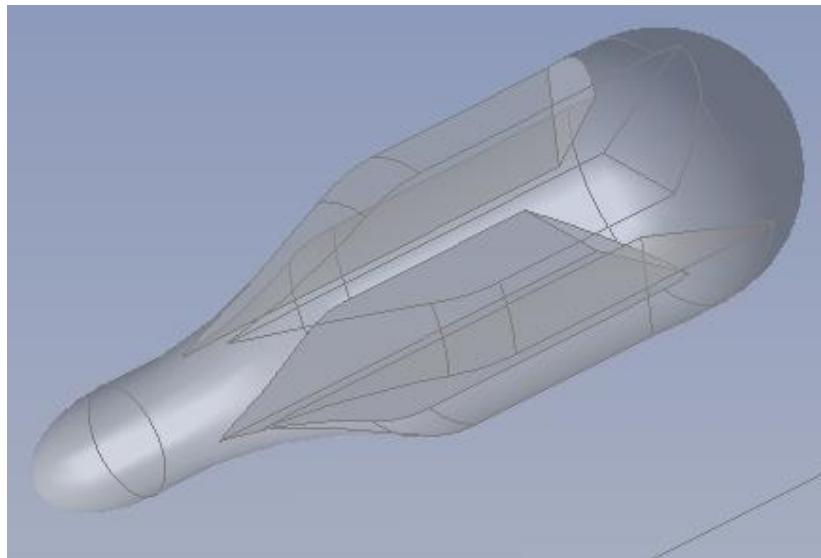


Illustration [1]

This overview allows us to observe the main idea of this project, also see the fluted part, which will make the device rotate.

The objectives and sections of this project are:

- ✓ CFD simulation in Fluent (Ansys)

With the aim to performance an exact CFD study, we will start the simulation from a 2D simulation of the basic shape, then it turn to a 3D simulation of the basic shape and finally it will be studied the final shape, (fluted shape).

- ✓ Wind tunnel experiments.

The experimental results help us to make sure that we are having the proper results in the simulations.

- ✓ Theoretic calculations.

In order to have a reference, in some cases we will use the theory to calculate results.

We are not using all of these method for all of the results. But they are very useful because we do not have reference values, which inform us of the right path.

● Fluid Mechanics Fundaments

In the study of this project, it is going to take place a series of fluid mechanic concept. Therefore, it will be explained the study fluids in motion in this first chapter. First of all, it has to decide how examine a flowing fluid. For this, it has two different options:

- Study the motion of an individual particle or group of particles
- Study a region of space as fluid flows through it, which is the control volume.

Finally, it will be chosen the second option because it has widespread practical application rather than how an individual fluid particles behave.

Physical Interpretation:

To cover the study fluid in motion by defining a control volume, it needs to develop a basic equations in integral form base on Basic Laws.

In this section it introduces a formula that we can use to convert the rate of change of any extensive property 'N' of a system to an equivalent formulation for use with a control volume.

$$\left[\frac{dN}{dt} \right]_{system} = \frac{\partial}{\partial t} \int_{CV} \eta \rho dV + \int_{CS} \eta \rho \vec{V} d\vec{S} \quad [1]$$

$\left[\frac{dN}{dt} \right]_{system}$; Rate of change of the system extensive property N.

$\frac{\partial}{\partial t} \int_{CV} \eta \rho dV$; Rate of change of the amount of property N in the control volume. The integral inside represents the instantaneous value of N in the control volume.

$\int_{CS} \eta \rho \vec{V} d\vec{S}$; Rate at which property N is exiting the surface of the control volume. The term $\rho \vec{V} d\vec{S}$ computes the rate of mass transfer leaving across control surface area element; multiplying by η computes the rate of flux of property N across the element; and integrating therefore computes the net flux of N out of the control volume.

This has been explained, it is going to be applied to physical laws.

- Conservation of mass. (Continuity equation)

The mass conservation says: The mass of the system remains constant.

$$\left[\frac{dM}{dt} \right]_{system} = 0$$

It sets for the general equation [1]:

$$N = M, \eta = 1$$

With this substitution, we obtain;

$$\left[\frac{dM}{dt} \right]_{system} = \frac{\partial}{\partial t} \int_{CV} \rho dV + \int_{CS} \rho \vec{V} d\vec{S}$$

Finally:

$$\frac{\partial}{\partial t} \int_{CV} \rho dV + \int_{CS} \rho \vec{V} d\vec{S} = 0 \quad [2]$$

$\frac{\partial}{\partial t} \int_{CV} \rho dV$; This term represents the rate of change of mass within the control volume.

$\int_{CS} \rho \vec{V} d\vec{S}$; This term computes the net rate of mass flux out through the control surface.

- Momentum Equation for International Control Volume.

In this section we are going to plug the Newton's second law in the general formula [1].

Please, take into account the system moving relative to an inertial coordinate system.

$$\vec{F} = \left[\frac{d\vec{P}}{dt} \right]_{system}$$

To derive the control volume formulation of Newton's second law, we set:

$$N = \vec{P} \text{ and } \eta = \vec{V}$$

Combining the Newton's second law with equation [1] for nonaccelerating control volume;

$$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t} \int_{CV} \vec{V} \rho dV + \int_{CS} \vec{V} \rho \vec{V} d\vec{S} \quad [3]$$

$\frac{\partial}{\partial t} \int_{CV} \vec{V} \rho dV$; Rate of change of momentum within the control volume, (the volume integral).

$\int_{CS} \vec{V} \rho \vec{V} d\vec{S}$; Net rate at which momentum is leaving the control volume through the control surface.

The momentum equation [3] is a vector equation. Here, it is shown the three scalar components, as measured in the xyz coordinates of the control volume;

$$\vec{F}_x = \vec{F}_{Sx} + \vec{F}_{Bx} = \frac{\partial}{\partial t} \int_{CV} u \rho dV + \int_{CS} u \rho \vec{V} d\vec{S}$$

$$\vec{F}_y = \vec{F}_{Sy} + \vec{F}_{By} = \frac{\partial}{\partial t} \int_{CV} v \rho dV + \int_{CS} v \rho \vec{V} d\vec{S}$$

$$\vec{F}_z = \vec{F}_{Sz} + \vec{F}_{Bz} = \frac{\partial}{\partial t} \int_{CV} w \rho dV + \int_{CS} w \rho \vec{V} d\vec{S}$$

- The Angular Momentum Principle.

To cover this section, it considers two different approaches, it can use an inertial (fixed) XYZ control volume or rotating xyz control volume.

It will use an inertial control volume. The aim of this is to develop a control volume equation for each of the basic physical laws.

The angular momentum is a meaningful concept for this study because the device of this project is going to experience an angular momentum, when this moves through a fluid.

The angular-momentum principle for a system in an inertial frame is:

$$\vec{T} = \left[\frac{d\vec{H}}{dt} \right]_{system}$$

Where \vec{T} = total torque exerted on the system by its surroundings.

\vec{H} = angular momentum of the system.

$$\vec{H} = \int_{M(system)} \vec{r} \times \vec{V} dm = \int_{V(system)} \vec{r} \times \vec{V} dV$$

The system equation must be formulated with respect to an inertial reference frame.

$$\vec{T} = \vec{r} \times \vec{F}_s + \int_{M(system)} \vec{r} \times \vec{g} dm + \overrightarrow{T_{shaft}}$$

\vec{F}_s ; Surface force exerted on the system.

$\int_{M(system)} \vec{r} \times \vec{g} dm$; Torque in the system.

$\overrightarrow{T_{shaft}}$; Torque at the shaft.

Let it use the general formula:

$$\left[\frac{dN}{dt} \right]_{system} = \frac{\partial}{\partial t} \int_{CV} \eta \rho dV + \int_{CS} \eta \rho \vec{V} d\vec{S}$$

Where;

$$N_{system} = \int_{M(system)} \eta dm$$

If it set $N = \vec{H}$, and $\eta = \vec{r} \times \vec{V}$, then:

$$\left[\frac{d\vec{H}}{dt} \right]_{system} = \frac{\partial}{\partial t} \int_{CV} \vec{r} \times \vec{V} \rho dV + \int_{CS} \vec{r} \times \vec{V} \rho \vec{V} d\vec{S}$$

If it is combined with the system equation with respect to an inertial reference frame, it gets the next:

$$\vec{r} \times \vec{F}_s + \int_{M(system)} \vec{r} \times \vec{g} dm + \overrightarrow{T_{shaft}} = \frac{\partial}{\partial t} \int_{CV} \vec{r} \times \vec{V} \rho dV + \int_{CS} \vec{r} \times \vec{V} \rho \vec{V} d\vec{S}$$

So far the system and control volume coincide at time t_0 ;

$$\vec{T} = \overrightarrow{T_{CV}}$$

Finally:

$$\vec{r} \times \vec{F}_s + \int_{CV} \vec{r} \times \vec{g} dm + \overline{T_{shaft}} = \frac{\partial}{\partial t} \int_{CV} \vec{r} \times \vec{V} \rho dV + \int_{CS} \vec{r} \times \vec{V} \rho \vec{V} d\vec{S} \quad [4]$$

The left side is an expression for all the torques that act on the control volume. Terms on the right express the rate of change of angular momentum within the control volume and the net rate of flux of angular momentum from the control volume.

It has to take into account that all velocities are measured relative to the fixed control volume.

Propellers

Propellers are a fluid machinery that is brought up in this project due to the importance of this type of turbine with regard to the device in this project. It can be seen in illustration [2] and [3] respectively.



Illustration [2]

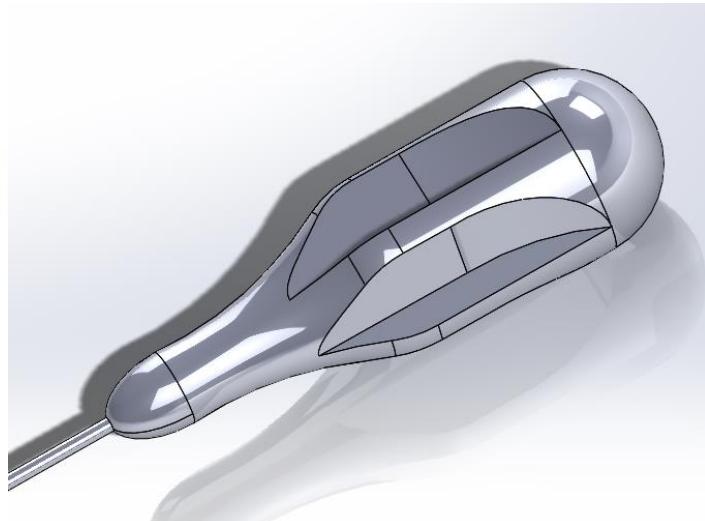


Illustration [3]

The main concept of this device is that a propeller produces thrust by imparting linear momentum to a fluid. Thrust production always leaves the stream with some kinetic energy and angular momentum that are not recoverable, so the process is never 100 percent efficient.

In the figure [4], it illustrates a uniform flow and in the new coordinates the flow is steady. The actual propeller is replaced conceptually by a thin actuator disk, through the actuator flow speed is continuous but pressure rises abruptly.

This figure does not show the swirl velocities that result from the torque required to turn the propeller.

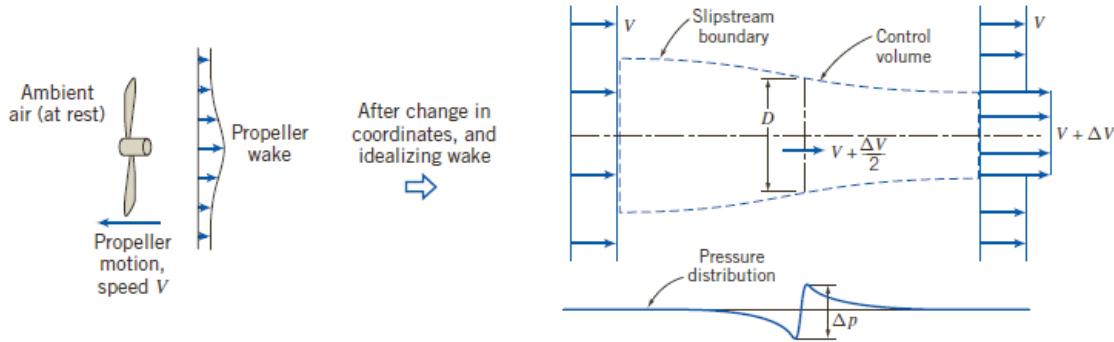


Illustration [4]

Let it analyze, with application of linear momentum in the axial direction, using a finite control volume, provides overall relations among slipstream speed, thrust, useful power output and minimum residual kinetic energy in the slipstream.

By using continuity and momentum equations in control volume form. The thrust produced is:

$$F_T = \dot{m} \Delta V \quad (\text{Constant thrust}) \quad \dot{m} \text{ mass flow [kg/s]}$$

And the power produced:

$$\mathcal{P}_{useful} = F_T V = \dot{m} V \Delta V$$

For incompressible flow, with negligible friction and heat transfer, the energy equation indicates that the minimum required input to the propeller is the power required to increase the kinetic energy of the flow, which may be expressed as:

$$\mathcal{P}_{input} = \dot{m} \left[\frac{(V + \Delta V)^2}{2} - \frac{V^2}{2} \right] = \dot{m} \left[\frac{2V\Delta V}{2} - \frac{(\Delta V)^2}{2} \right] = \dot{m} V \Delta V \left[1 + \frac{\Delta V}{2V} \right]$$

Propulsive efficiency can be written as:

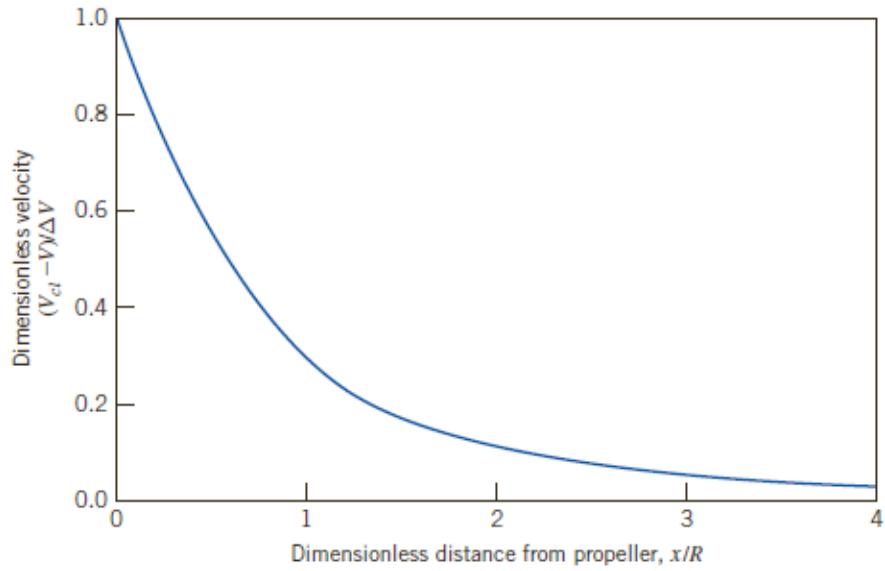
$$\eta = \frac{\mathcal{P}_{useful}}{\mathcal{P}_{input}} = \frac{1}{1 + \frac{\Delta V}{2V}}$$

It can increase the efficient by increasing V. At constant thrust, ΔV can be reduced if \dot{m} is increased.

In equation below $V_{cl}(x)$ is the centerliner velocity at location x upstream of the disk, while V is the upstream velocity.

$$V_{cl}(x) = V + \Delta V \left(1 - \frac{x}{\sqrt{x^2 + R^2}} \right)$$

Let it see the relationship in plot [1].



Plot [1]

The plot shows that the effects of the propeller are only felt at distances with two radii of the actuator disk.

To calculate the interaction between a propeller blade and the stream and therefore to determine the effects of blade aerodynamic drag and the stream and therefore to determine the effects of blade aerodynamic drag on the propeller efficiency.

In the illustration [6] it can see a diagram of blade element and relative flow velocity vector. Illustration [5] helps to visualize the figure next to this.

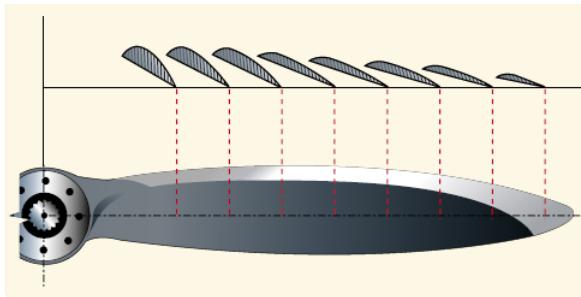


Illustration [5]

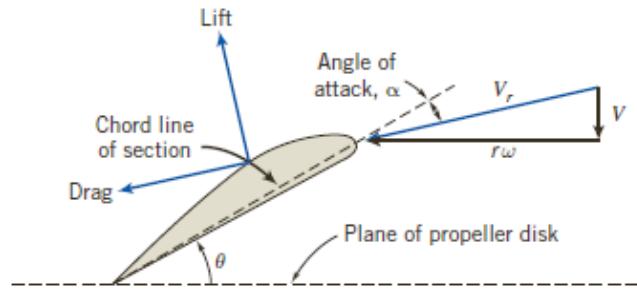


Illustration [6]

θ ; Angle to the plane of the propeller disk.

dr ; Thickness (into the plane of the page).

V_r ; Relative velocity vector

\emptyset ; Effective pitch angle (Angle that V_r makes with the plane of the propeller disk).

V ; Speed of advance

$r \omega$; Blade peripheral speed.

It finds the magnitude of the resultant dF_r parallel to the velocity vector \vec{V} :

$$dF_T = dL \cos \phi - dD \sin \phi = q_r c dr (C_L \cos \phi - C_D \sin \phi)$$

$$q_r = \frac{1}{2} \rho V_r^2$$

q_r ; Dynamic pressure.

c ; Chord length.

C_L & C_D ; Lift and drag coefficient.

It can also generate an expression for the torque that must be applied to the propeller:

$$dT = r(dD \cos \phi + dL \sin \phi) = q_r r c dr (C_D \cos \phi + C_L \sin \phi)$$

These two expressions may be integrated to find the total propulsive thrust and torque, assuming N independent blades mounted on the rotor:

$$F_T = N \int_{r=R_{hub}}^{r=R} dF_T = q N \int_{R_{hub}}^R \frac{(C_L \cos \phi - C_D \sin \phi)}{\sin^2 \phi} c dr \quad [5]$$

$$T = N \int_{r=R_{hub}}^{r=R} T = q N \int_{R_{hub}}^R \frac{(C_D \cos \phi + C_L \sin \phi)}{\sin^2 \phi} r c dr \quad [6]$$

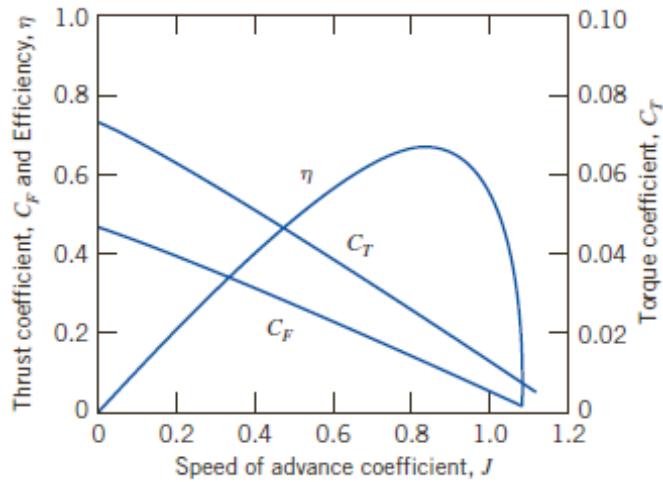
This two above equations are not used, and propeller performance characteristics usually are measured experimentally.

$$J \equiv \frac{V}{n D}$$

n ; Rotational speed. [r.p.s.]

J ; Speed of advance coefficient.

Plot [2] shows typical measured characteristics for a marine propeller.

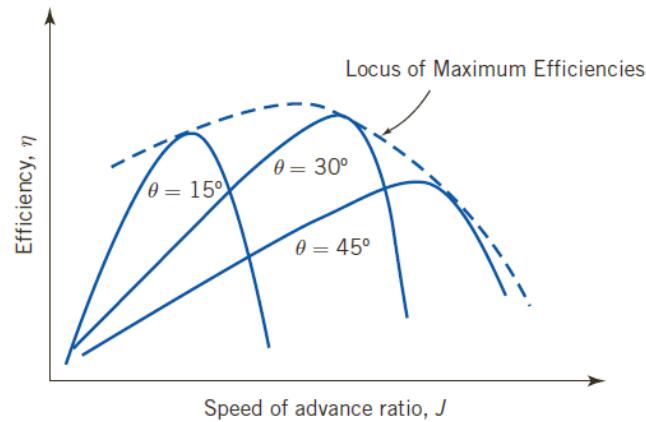


Plot [2]

Dependent variables are the thrust coefficient, C_F , the torque coefficient, C_T , the power coefficient, C_P , and the propeller efficiency, η , defined as

$$C_F = \frac{F_T}{\rho n^2 D^4} \quad C_T = \frac{T}{\rho n^2 D^5} \quad C_P = \frac{\mathcal{P}}{\rho n^3 D^5} \quad \text{and} \quad \eta = \frac{F_T V}{\mathcal{P}_{input}}$$

In order to improve performance, some propellers are designed with variable pitch. Plot [3].



Plot [3]

θ ; Pitch angle.

Dimensional Analysis:

The potential of Dimensional Analysis is very meaningful to this project. This physic concept allows to express reliable results and makes testing on a wind tunnel, which has different parameters with regard to real. With this aim it will use dimensionless number (Reynolds number).

To reach this goal, it will take into consideration 'Buckingham Pi Theorem'.

To explain this section, let it consider a sphere, which depends on the diameter D, fluid density ρ , viscosity μ and fluid speed V. For this problem it wants to obtain the drag F.

$$F = F(D, \rho, \mu, V)$$

The nature of function:

$$g(F, D, \rho, \mu, V) = 0$$

Where g is an unspecified function. The Buckingham Pi theorem states that we can transform a relationship between n parameters of the form.

$$g(q_1, q_2, q_3 \dots, q_n) = 0$$

Into a corresponding relationship between $n-m$ independent dimensionless Π parameters in the form.

$$G(\Pi_1, \Pi_2, \Pi_3 \dots, \Pi_{n-m}) = 0$$

$$\Pi_1 = G(\Pi_2, \Pi_3 \dots, \Pi_{n-m})$$

m ; It is usually the minimum number.

r ; Number of independent dimensions. [Mass, length, time...] ($m \neq r$)

$q_1, q_2, q_3 \dots, q_n$; Parameters.

Let it focus on the sphere problem:

$$g(F, D, \rho, \mu, V) = 0 \quad \text{or} \quad F = F(D, \rho, \mu, V)$$

Leads to

$$G\left(\frac{F}{\rho V^2 D^2}, \frac{\mu}{\rho V D}\right) = 0 \quad \text{or} \quad \frac{F}{\rho V^2 D^2} = G_1\left(\frac{\mu}{\rho V D}\right)$$

The independent dimensionless Π parameters must be determined experimentally. The ($m \neq r$) dimensionless Π parameter is not independent if it can be formed from any combination of one or more of the other Π parameters.

- Determining the Π group.

Before starting choosing parameters, we might expect that some parameters are going to take place in our experiment. In this case it should include these parameters. When the experiment is over, it will see if our prediction was right or wrong.

The steps listed below is a recommended procedure for determining the Π parameters:

1. List all the dimensional parameters involved.
2. Select a set of fundamental (primary) dimensions.
3. List the dimensional of all parameters that includes all the primary dimensions.
4. Select a set of r dimensional parameters that includes all the primary dimensions.
5. Set up dimensional equations, combining the parameters selected in Step 4 with each of the other parameters in turn, to form dimensionless groups.
6. Check to see that each group obtained is dimensionless.

The functional relationship among the Π parameters must be determined experimentally.

For this experiment it would like to take into consideration these dimensionless parameters:

$$Re = \frac{\rho V D}{\mu} = \frac{V D}{\nu} \quad \text{Reynolds number.}$$

$$Eu = \frac{\Delta p}{\frac{1}{2} \rho V^2} \quad \text{Euler number.}$$

$$Ca = \frac{p - p_v}{\frac{1}{2} \rho V^2} \quad \text{Cavitation number. } p_v(\text{Pressure of cavitation})$$

$$Fr = \frac{V}{\sqrt{gL}} \quad \text{Froude number.}$$

$$Eu = \frac{L \rho V^2}{\sigma} \quad \text{Weber number.}$$

$$M = \frac{V}{c} \quad \text{Mach number.}$$

Methods.

This section contains an overview of the experimental and analytical procedure.

Design of a rotatory device.

Pre-study:

Before starting our simulation, it has to be done a pre-study. It needs to do this because the results from the simulation must match in coherent and logical way.

For this purpose, it is going to be presented the design 2D and 3D design model of the device in this section. The main goal of this project is the study of drag coefficient and the rotating behavior as well. For accomplishing the drag coefficient in the simulations, it will compare the specimens shape with other elemental shapes. By this comparison, we will be able to predict logical results in the simulation.

In the next page, it is introduced the device design, which is based on MiniPAT-348A of WILDLIFE COMPUTERS company. This last MiniPAT can be seen below.



Illustration [7]. WILDLIFE COMPUTERS

In the illustration [8] it can be found a preview of the geometry of the device. Please, the reader must take into consideration that all the geometries draws are added in the annex of this project.

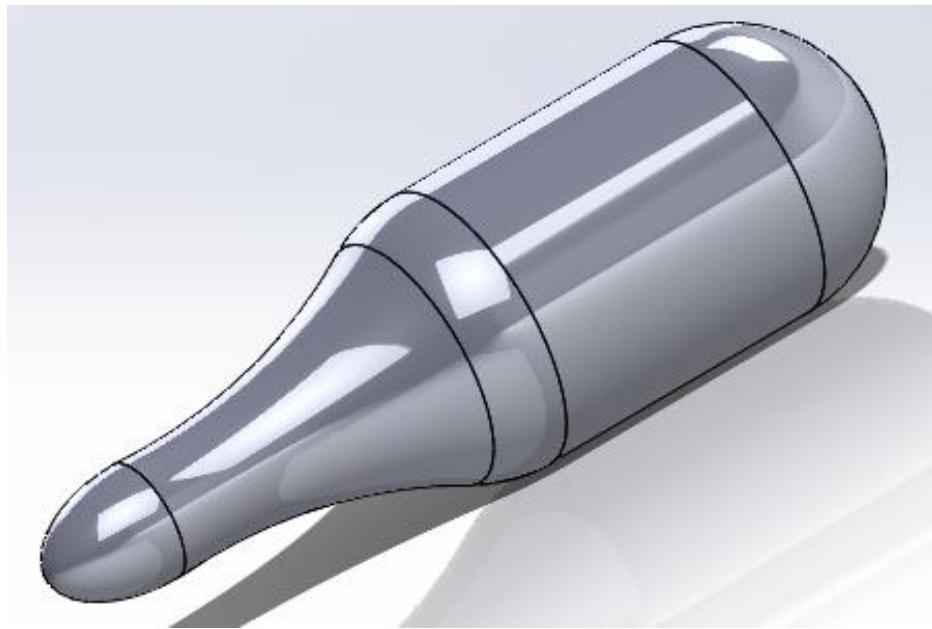


Illustration [8].

So as has been said, it will launch the 2-D simulation and 3-D simulation too.

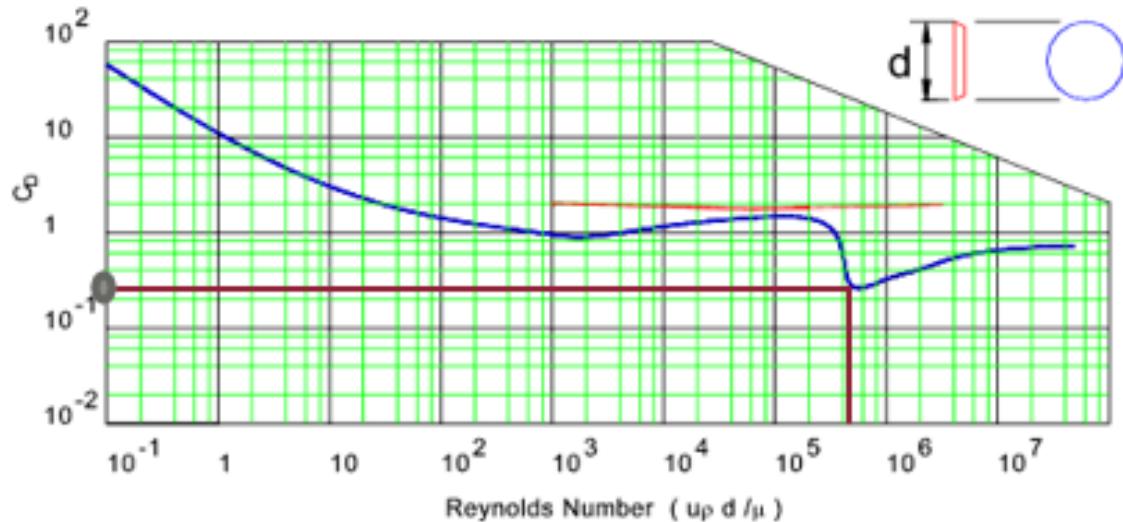
The main goals in the 2-D simulation is to see what a proper mesh is and compare it to an approximated shape. We also can see the velocity vectors, pressure lines, stream lines... pretty clear, because the 3-D model is a revolution geometry that fulfills symmetry of the 2-D models.

It is going to use experimental graphs, where the drag coefficient is as a function of the Reynolds number. For this comparison we have two graph, the 2D reference geometry will be a cylinder and the 3D geometry will be a sphere. Below we can see these graphs.

For both Fluent simulations, we are going to have a Reynolds number of 246813. In the next two plots we are going to check the drag coefficient of a cylinder and sphere. These basic geometries are going to be used as reference geometries for the initials shapes.

2D

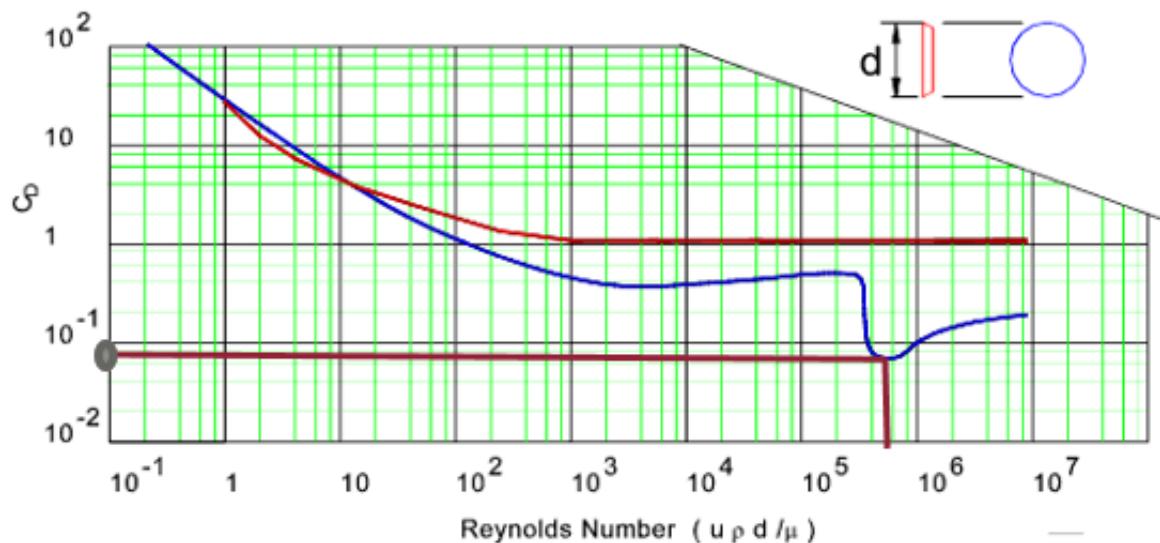
Drag for Infinitely long **PLATE** and infinitely long **CYLINDER**



Plot [4]

3D

Drag for **DISC** and **SPHERE**



Plot [5]

Design of the MiniPAT.

With the aim to clarify to the reader of this project how the non-fluted and fluted device looks like, we have attached a couples of illustration in this section. These illustration were design in SOLIDWORKS.

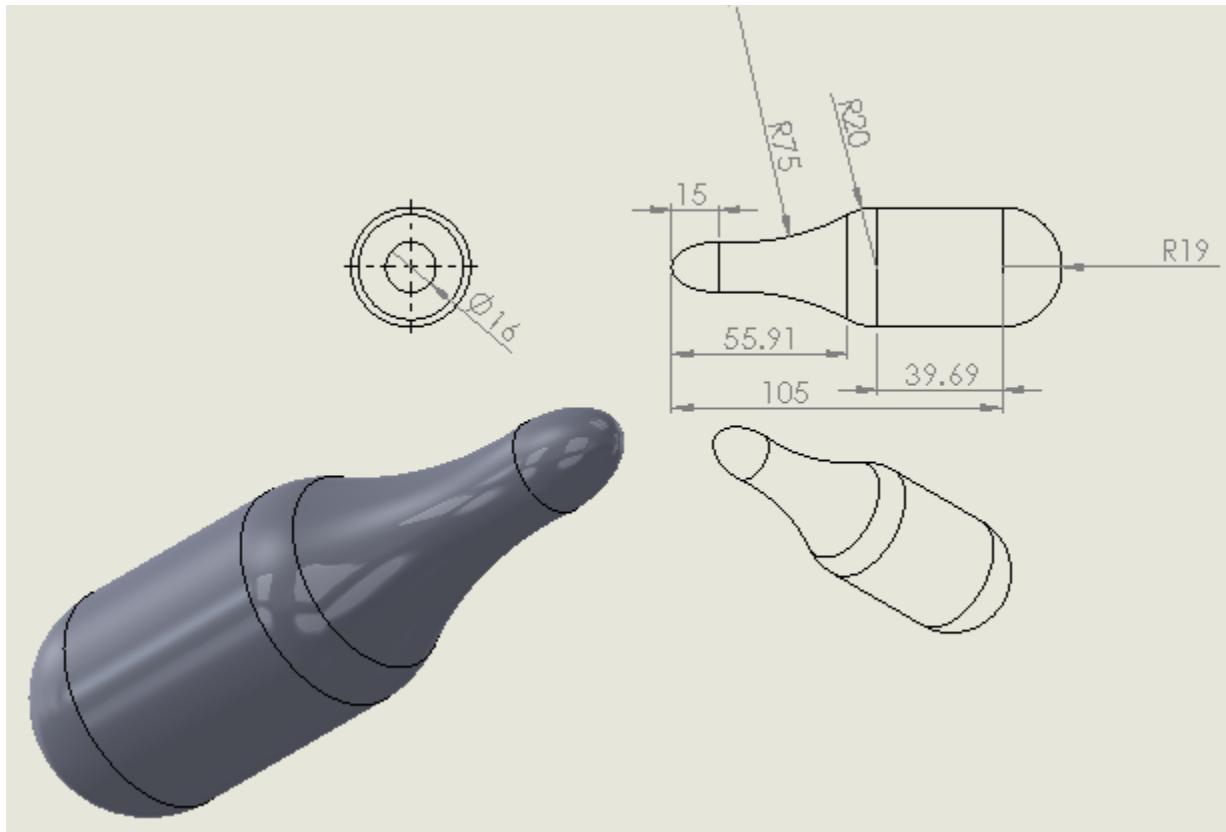


Illustration [9].

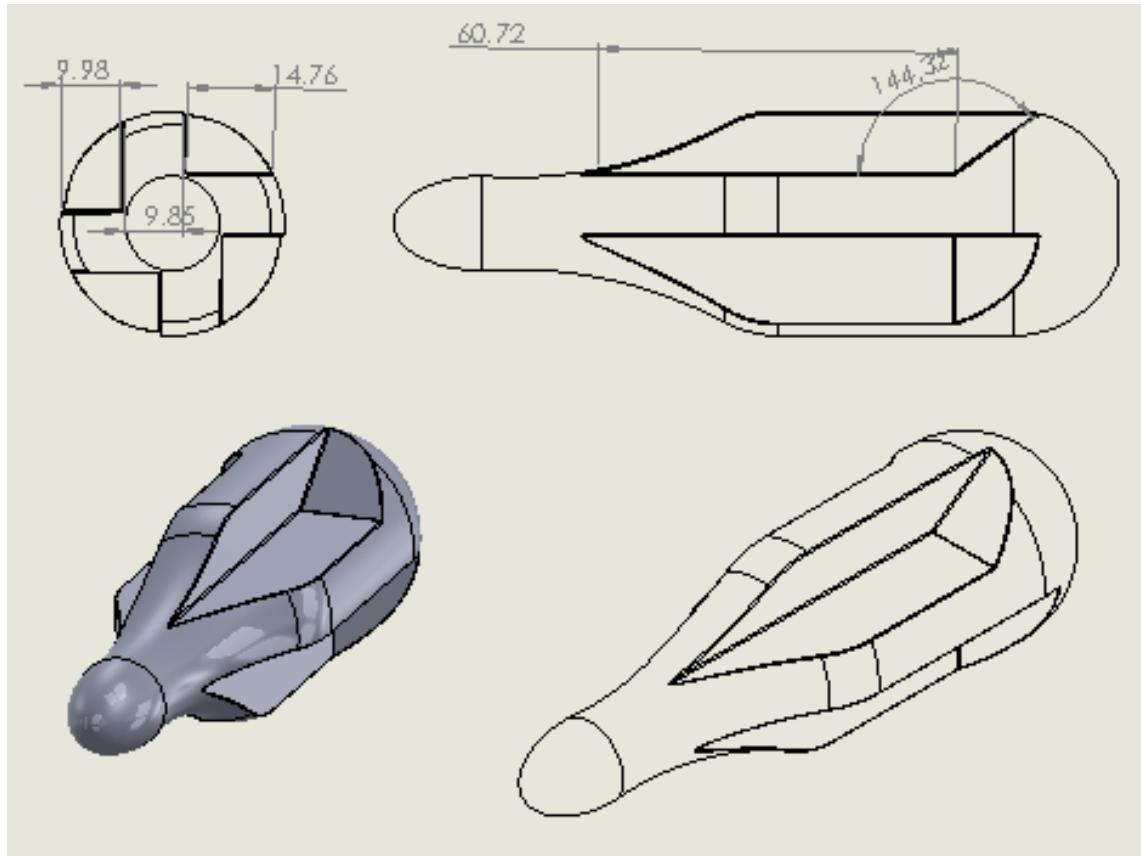


Illustration [10].

Simulation. Ansys (Fluent).

With the aim to develop a good simulation, it is going to focus on a proper mesh that is why we have to design different mesh and validate the most optimum mesh because depending on the mesh we will have more or less accurate results.

The actual reason why we need to launch different simulation with different meshes is because we do not have a reference for this experiment. This method is called independent study.

Let's start this section with the 2D simulation.

Two Dimensional simulation.

Before starting this section I would like to get across of an important point.

A 2D simulation in fluent is not really consider as a 2D dimension. Once the geometry is exported in Ansys as a plane, Ansys gives a thickness of unit valor to the geometry, this is the reason

and as it has been said, we are going to compare the 2D problem with a cylinder. In the illustration [11], this point can be clarified.

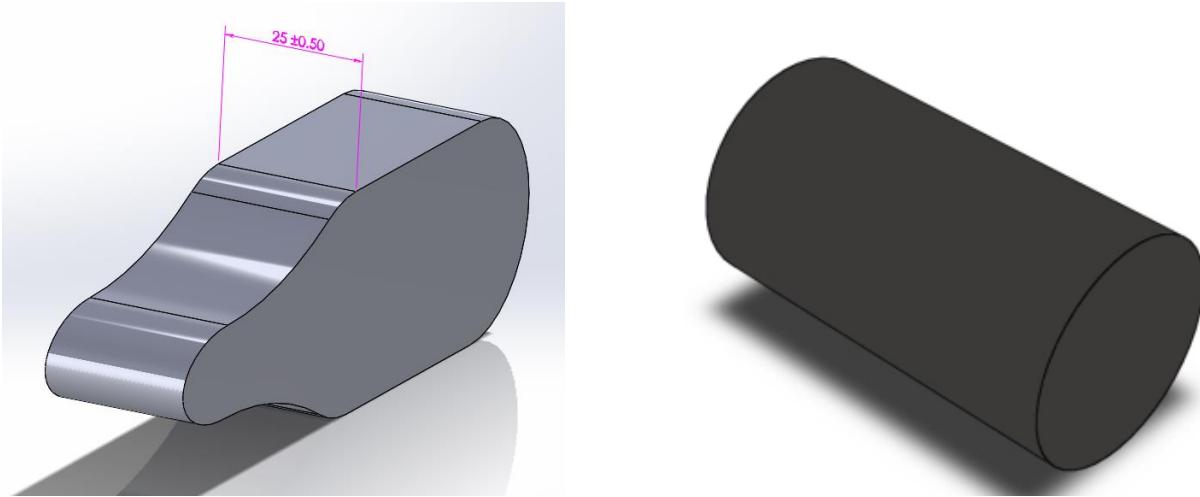


Illustration [11].

In the table below, it can be find the setting for the mesh and a briefly explanation about why this setting was chosen.

Settings	Selections	Justification
Solver Preference	Fluent	It has been selected Physics Preference CFD obviously, this mesh is used for CFD problems, with a relevance of 0.
Units	Metrics (m,Kg,N,s,V,A)	This units are due to the parameters used to calculate the drag coefficient that is why it considers [m] the most suitable unit. On the other hand, it is advisable the use of international units.
Method	None	It has been tested all Method's options and finally none is consider appropriate, because we do not mind the Method, this alternative option (none) allows better adaptation in the meshing.

Sizing	Proximity and Curvature Edge Sizing Smoothing	It provides suitable minimum and maximum cells. The minimum size is 0.0005 m and the maximum 0.2 m. This command make us improve the distribution of cells along the shape. High
Inflation	Local inflation	Global Inflation is not available for 2D geometries such as this. In this inflation it was used an inner and upper boundary. This command allows us to modify and improve the meshing at any selected location.
Orthogonal Quality	0.94178	It has to be close to 1, being no important the high aspect ratio cells.
Nodes/Cells	56850/97832	I consider an appropriate amount, which are not going to take a long time to run on.
Growth Rate	Sizing 1.05 Inflection 1.2	I select 1.05 because I do not consider very important the domains far from the airfoil. In the inflection I set up the growth rate to solve the boundary layer.
Named Selections	4 Named Selections	Inlet. Wall. Outlet. Turbine.

Table [1].

After all these settings this is results:

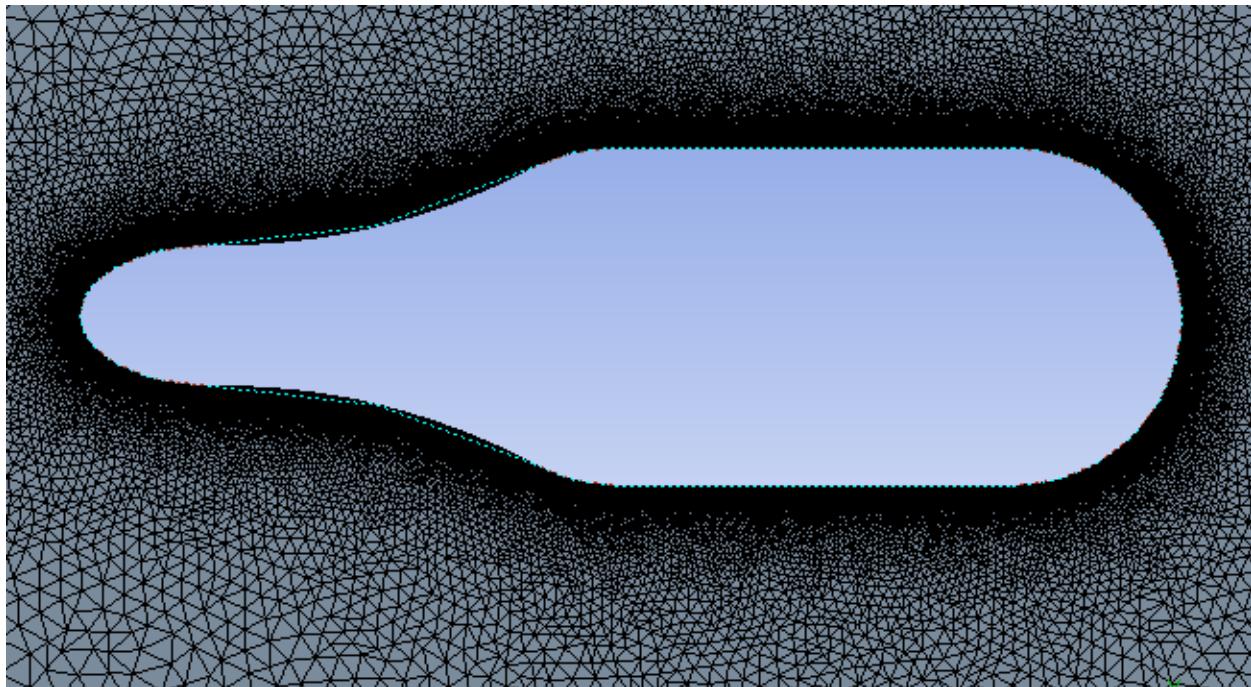


Illustration [12].

One important consideration on the meshing process is the proper setting of the boundary layer, for this, it has been calculated the boundary layers on the website: <http://www.pointwise.com/yplus/>. In this part, I want to introduce the y^+ , which is an indicator about the mesh is able to resolve the boundary layer properly.

$$y^+ \equiv \frac{u_* y}{\nu}$$

u = friction velocity.

y = distance from wall

ν = dynamic viscosity

Input

<input type="button" value="Reset to Sea Level Conditions"/>		
$U_\infty:$	2	freestream velocity (m/s)
$\rho:$	998.2	freestream density (kg/m ³)
$\mu:$	0.001003	dynamic viscosity (kg/m s)
$L:$	0.124	reference length (m)
$y^+:$	1.4	desired y^+

Output

Compute Wall Spacing	
$\Delta s:$	0.000014975416039808 wall spacing (m)
$Re_x:$	246813.1605184447 Reynolds number

Illustration [13].

It is important that the mesh near wall is properly sized to ensure accurate simulation of the flow-field. This calculator above computes the height of the first mesh cell off the wall required to achieve a desired Y_+ using flat-plate boundary layer theory.

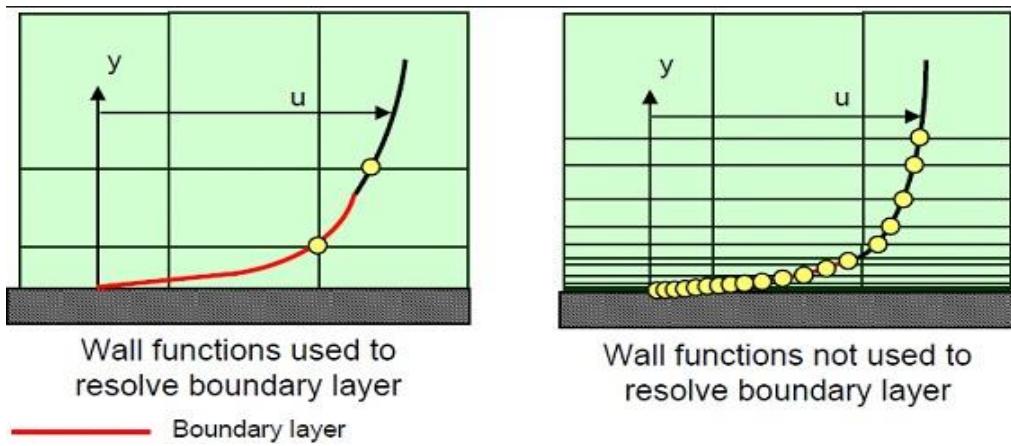


Illustration [14].

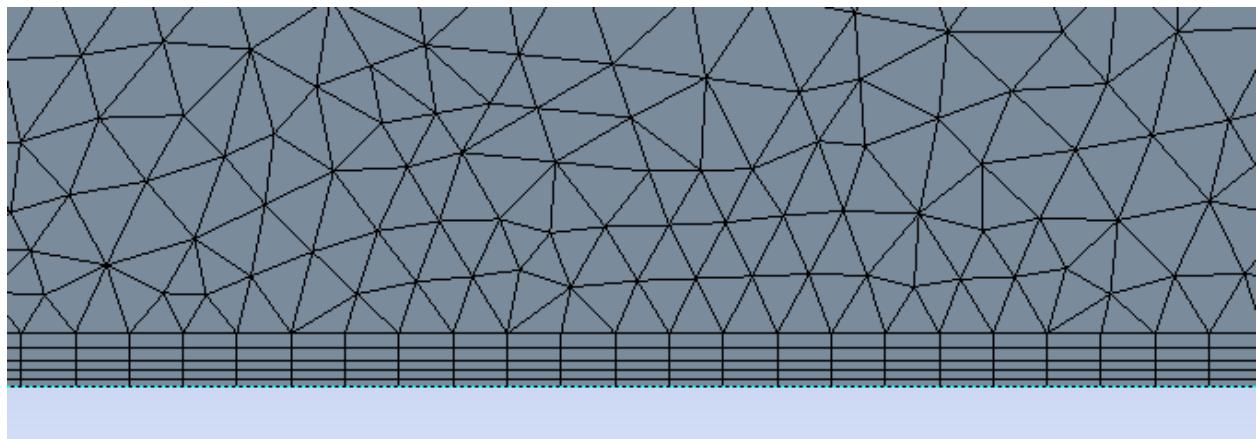


Illustration [15].

We can appreciate the first wall spacing here is about 10^{-5} by the scale pattern.

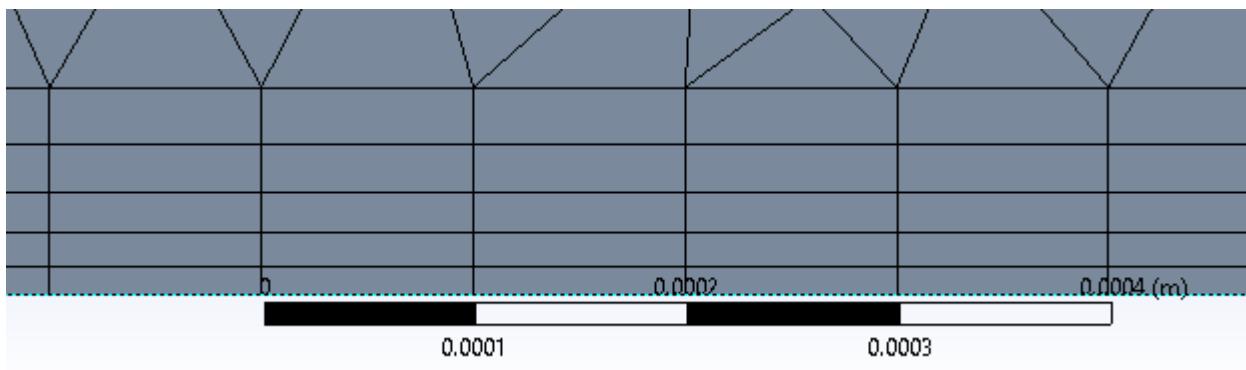


Illustration [16].

With the aim to clarify the Fluent model setting, below there is a briefly explanation about why this setting was chosen.

Settings	Selections	Justification
Solver Type	Density-Based	The density-based solver solves the governing equations of continuity, momentum, and energy and species transport simultaneously, this is how we have considered this problem. No turbulence regimen. I will do some specifications.
Time Scheme	Steady	The airfoil is going to be considered stopped and the fluid field is moving through the airfoil. As a wind tunnel experiment. There are no needs to do this study as transient.
Models	Viscosity	Spalart- Amaras (1 eqn).No Turbulent viscosity. This model is really useful for all 2D simulations, also well used for aerodynamics problems. It is easy to make it converge.
Fluid Material	Liquid Water	It is selected an incompressible flow. The Fluid material is take it from the Data-Base of fluent. The main properties of this is: Density of 998.2 Kg/m ³ Viscosity of 0.001003 kg/m-s
Airfoil Material	Aluminum	It is a good choice for this rotated device, because it is being immersed in water, avoiding corrosion and rusted issues. This is also a light material. Anyhow, the tests will be in plastic, but these make no significant differences.

Zone Conditions	Zone	Zone2dturbineshape_surface-plane1 set it as fluid (water-liquid)
Boundary Conditions	Inlet Zone2dturbineshape Outlet Turbine Walls	-Velocity inlet 2 m/s. -Interior. -Outflow. -Wall. -Symmetry. No thermal setting.
Reference Values	Inlet	That is why, we already set up this field. <i>'Reference values for velocity, density, temperature, etc. will update from the free-stream values as described on the previous slide'</i> That means that the calculations are going to start from the Fairfield to the rest of the meshing.
Solution Scheme	*Solution Methods	-Gradient : Least Squares Cell Based -Pressure : Second Order -Momentum : Second Order Upwind -Modified Turbulent Viscosity: First Order Upwind.
Monitors	Residuals, cd & cf.	I have plotted residuals, cd and cf. These are going to compare with experimental data. In the plots and prints we can see how the results converge. Lift coefficient must be about zero.
Initialization Type	Standard Initialization	This command make a simpler initialization starting from the far field. Standard initialization. Computed from Inlet.
Convergence Criteria	None	It has no convergence criteria, we can see if the convergence is appropriate or not from our own knowledge.

Table [2].

Three Dimensional simulation. Non-Fluted

For this section, we have done different meshes in order to find a good mesh. Here we present all of the meshes and at the end of the section we will select one of them and it will be described the settings and characteristics of this mesh. This is a method is called independent study, it is a tricky way to find a good mesh but having no references it is the only way to face this simulations.

Other drawback it the difficult Reynolds number region, where we are simulating in the plot C_d vs Reynolds number, where the plot is almost vertical, making this difficult to compare with elemental geometries.

The meshes start from a simple sample of it (low number of cells and non-refined locations) to a high complexity mesh.

Mesh N° One.

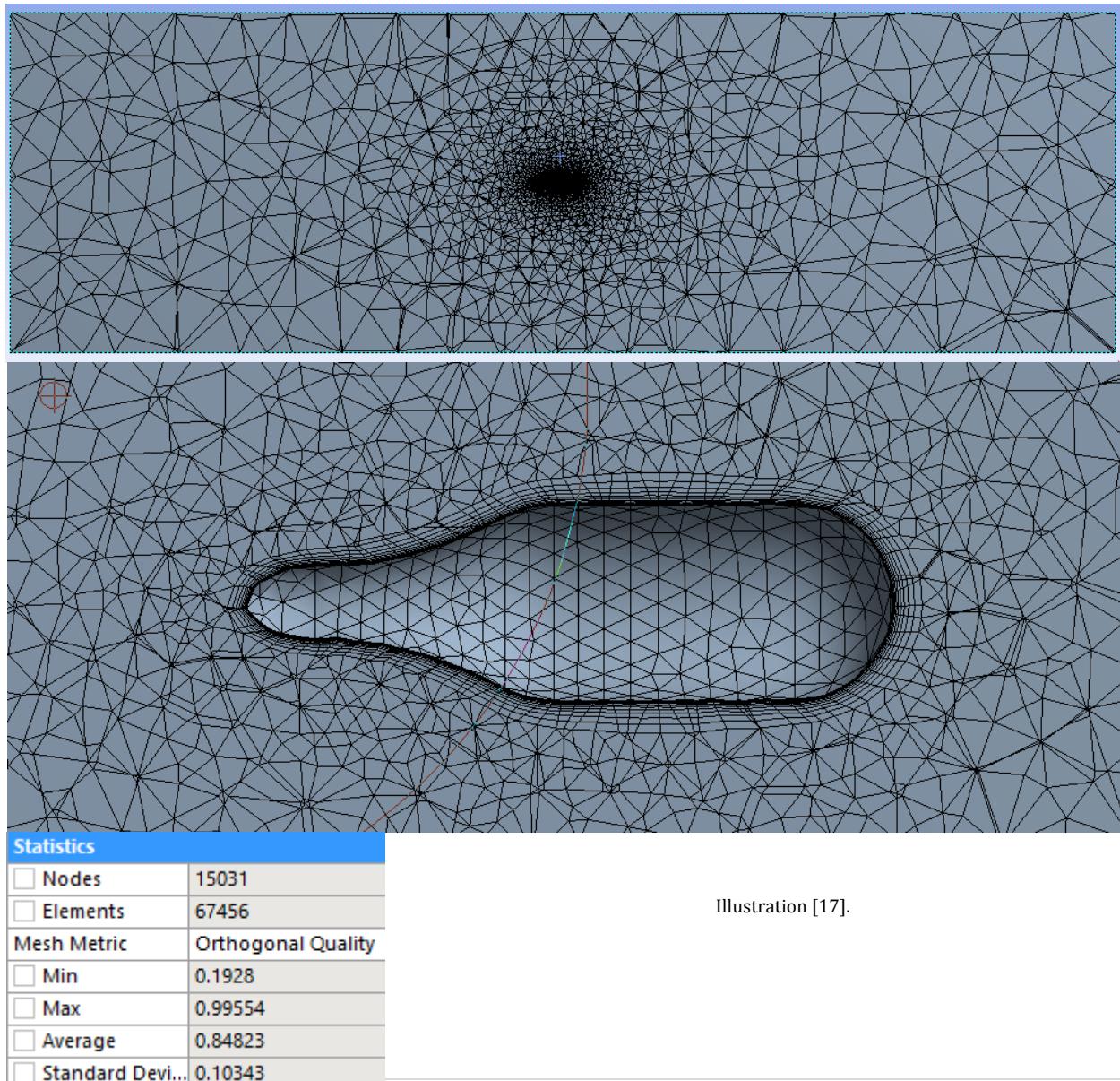
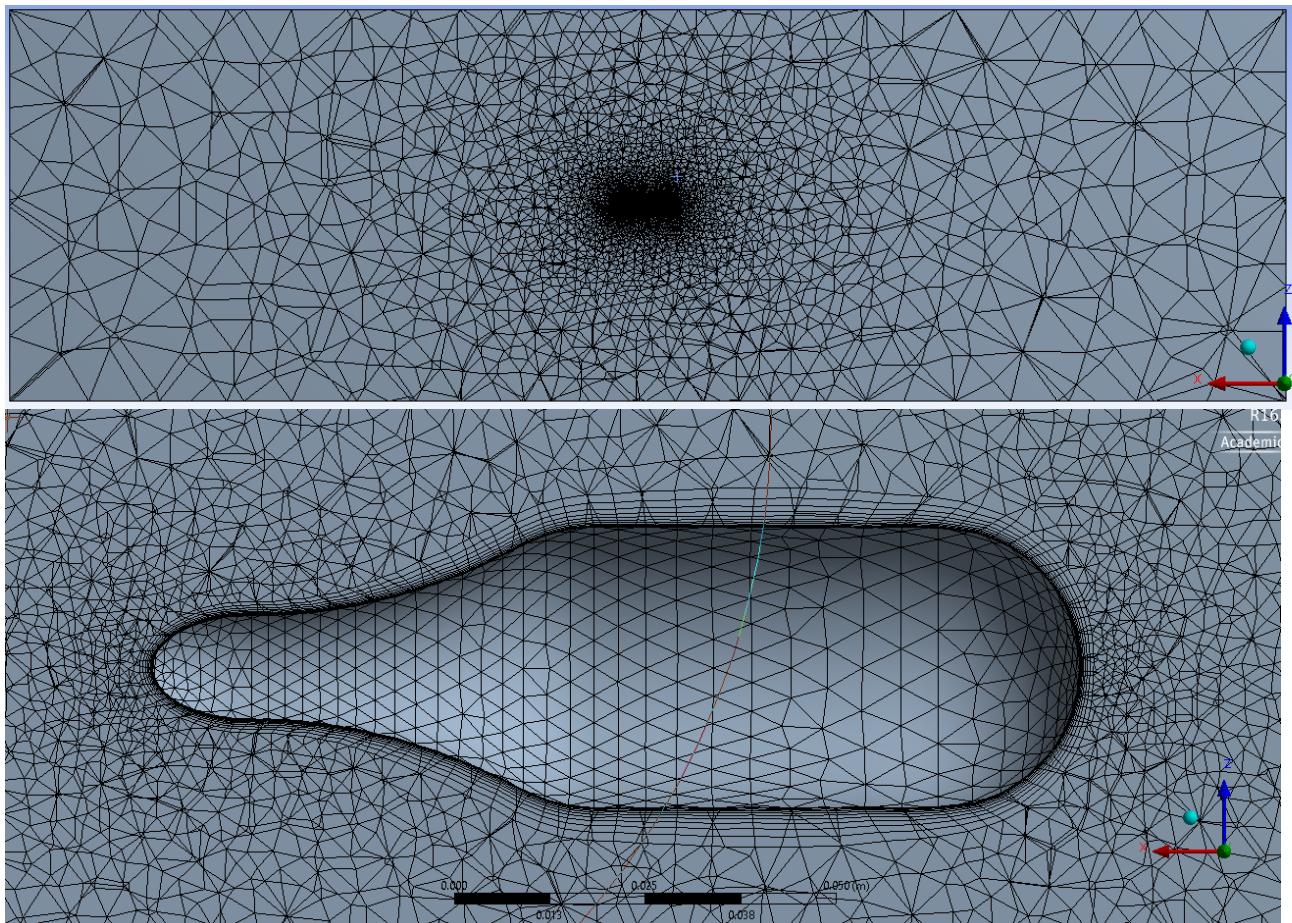


Illustration [17].

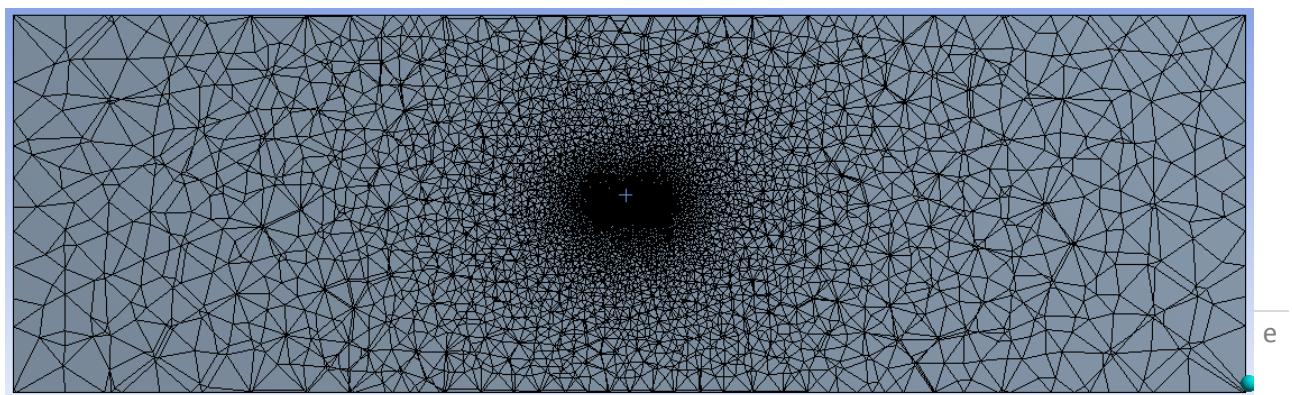
Mesh N° Two.

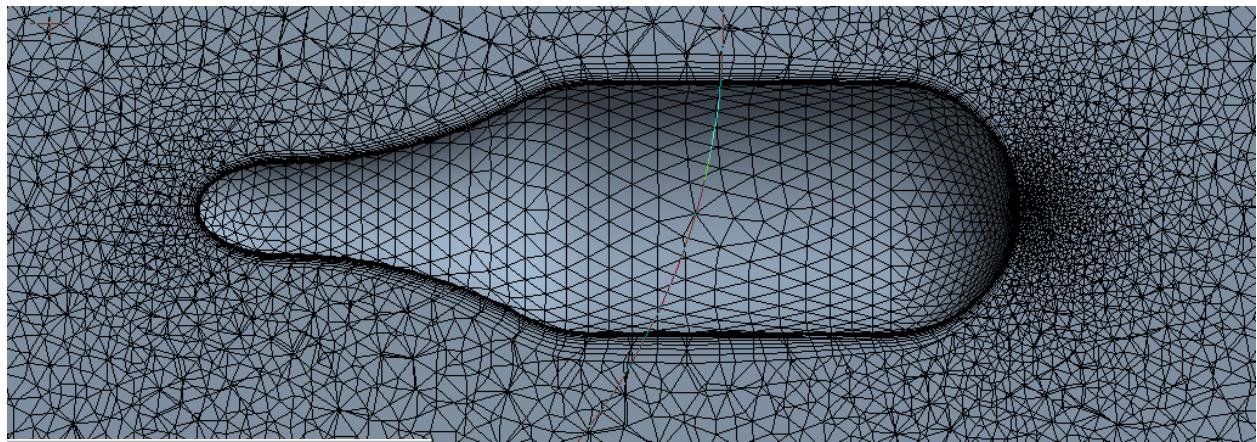


Statistics	
<input type="checkbox"/> Nodes	42159
<input type="checkbox"/> Elements	199738
Mesh Metric	Orthogonal Quality
<input type="checkbox"/> Min	0.25947
<input type="checkbox"/> Max	0.99903
<input type="checkbox"/> Average	0.86186
<input type="checkbox"/> Standard Devi...	9.5503e-002

Illustration [18].

Mesh N° Three.

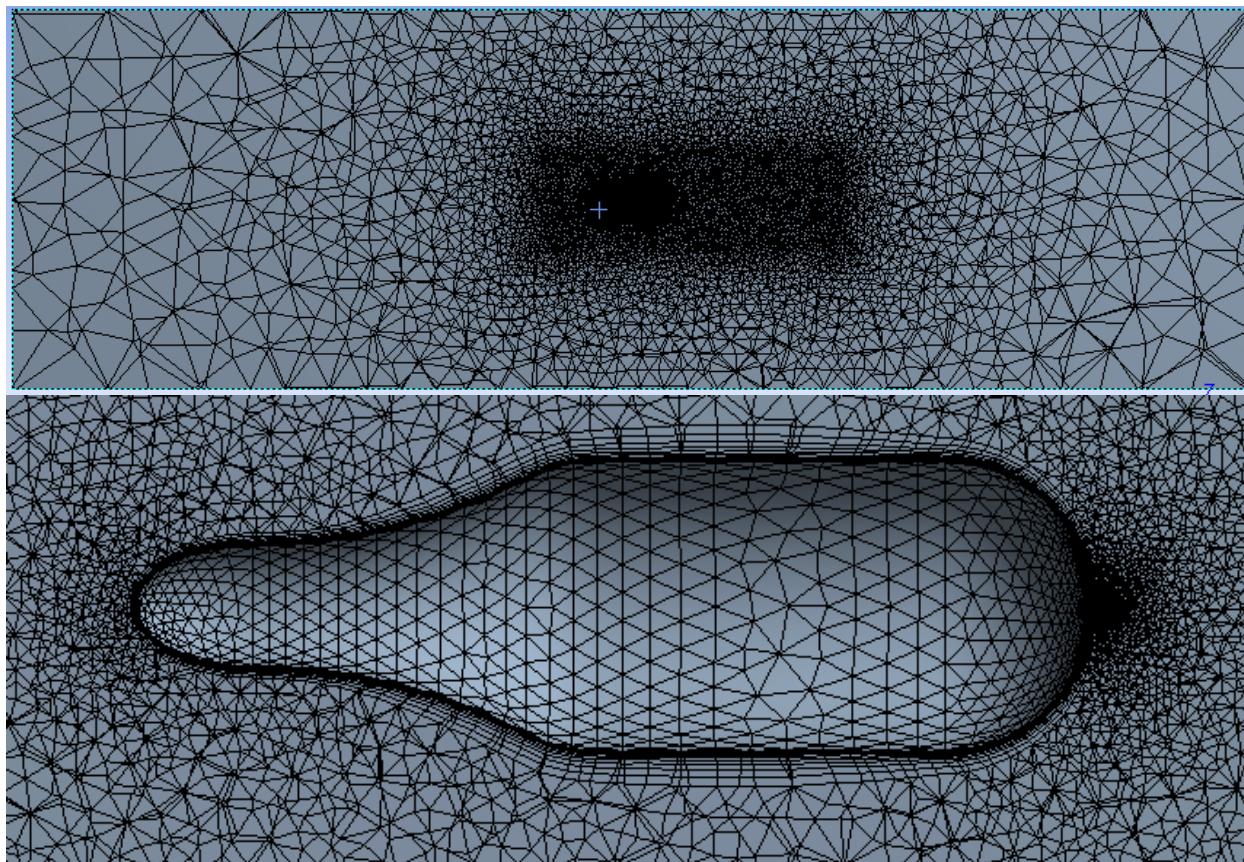




Statistics	
<input type="checkbox"/> Nodes	130753
<input type="checkbox"/> Elements	664208
Mesh Metric	Orthogonal Quality
<input type="checkbox"/> Min	0.27906
<input type="checkbox"/> Max	0.99993
<input type="checkbox"/> Average	0.87431
<input type="checkbox"/> Standard Devi...	8.5062e-002

Illustration [19].

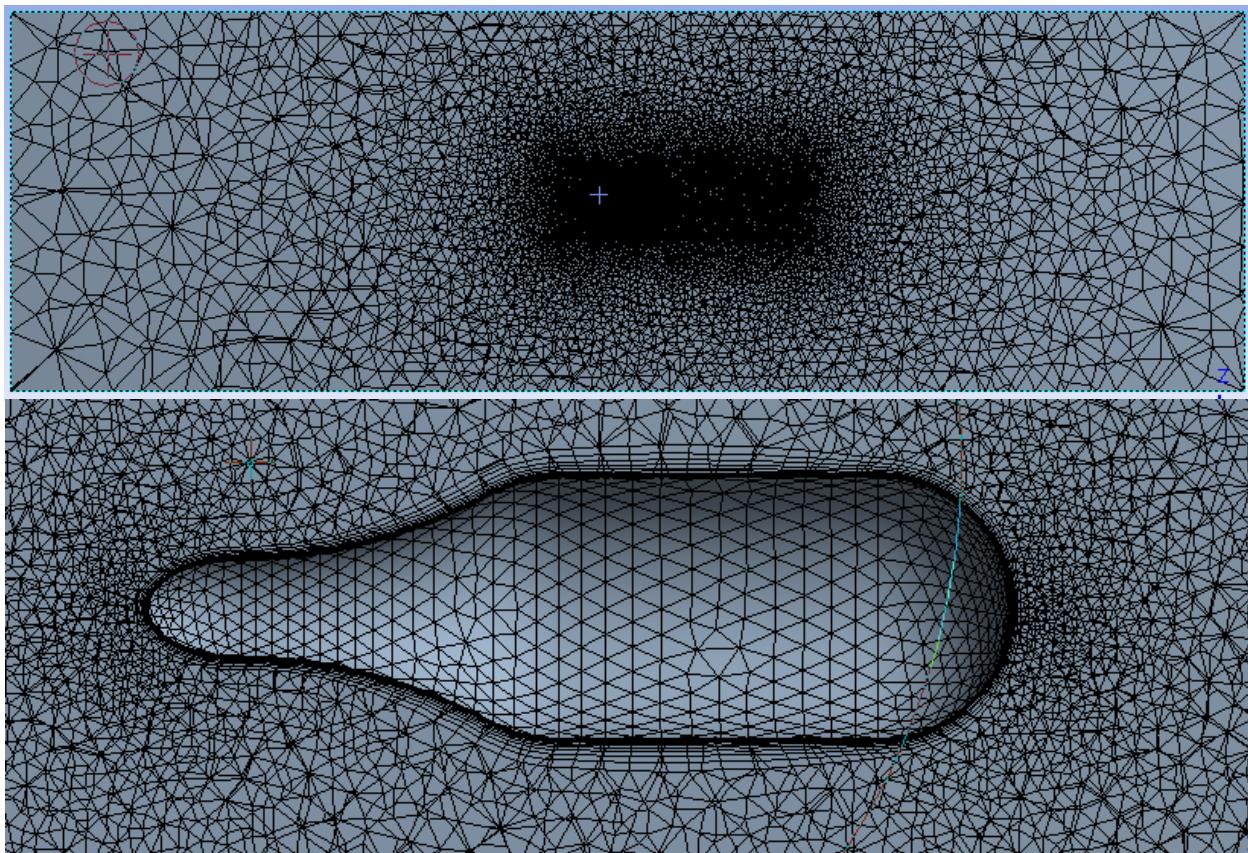
Mesh N° Four



Statistics	
Nodes	137495
Elements	726918
Mesh Metric	Orthogonal Quality
Min	0.25595
Max	0.99998
Average	0.87227
Standard Deviation	8.5468e-002

Illustration [20].

Mesh N° Five



Statistics	
Nodes	180729
Elements	1007196
Mesh Metric	Orthogonal Quality
Min	0.23707
Max	0.99901
Average	0.8715
Standard Devi...	8.4215e-002

Illustration [21].

After these five meshes in order to improve the aspect ratio and see if this concept really affect to the results, we have done two more meshes. See below.

Sixth Mesh (corrected aspect ratio)

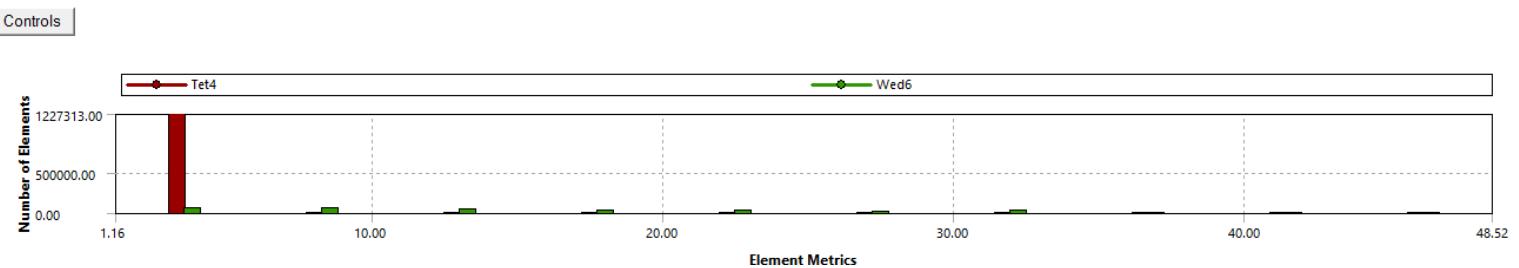
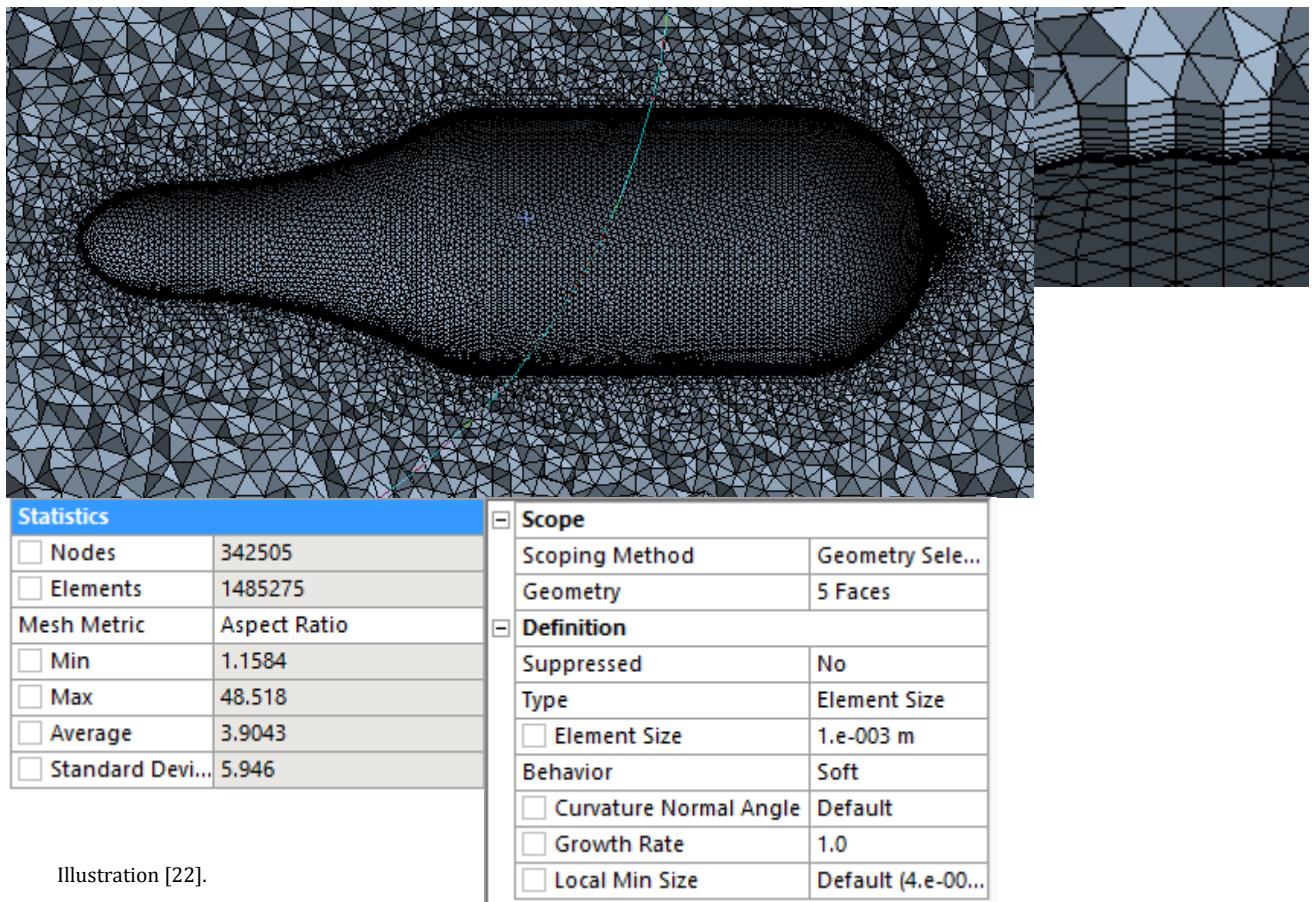


Illustration [23].

Seventh Mesh (corrected aspect ratio)

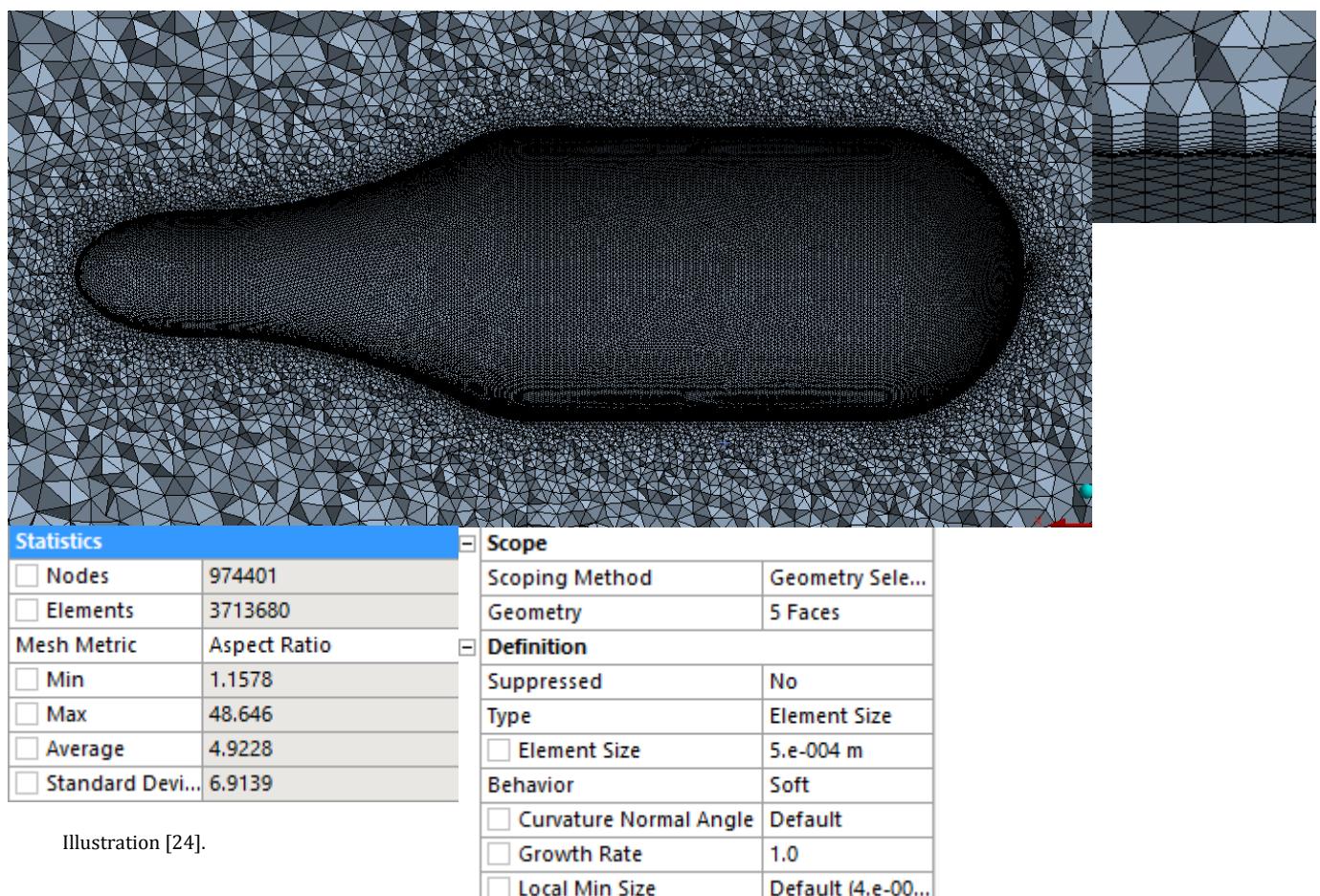


Illustration [24].

Controls

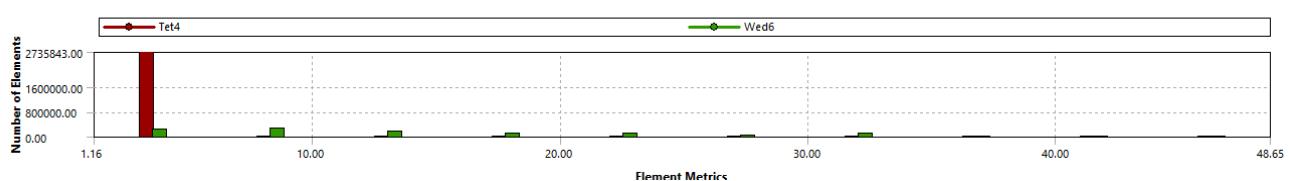


Illustration [25].

So, after all of these consideration, we have seen where the highest aspect ratio values take place. In the next five illustrations we are able to see the region of the mesh where the highest aspect ratio is.

Due to the goal to resolve the boundary layer properly, we need to set a wall height as small as necessary to solve the turbulent regimen which goes through the surface of the device. This flow are going to produce drag forces by shear forces, which will increase the drag coefficient. This is the main reason why we need to have a good mesh next to the surface.

With this aim, it is very important to keep the proper boundary layer solution, even when this value might increase the aspect ratio though.

Other offered solution is to reduce the aspect ratio, nevertheless the number of cells would incredibly rise up. In this case I strongly recommend to make sure your computer is powerful enough.

Aspect ratio of 13

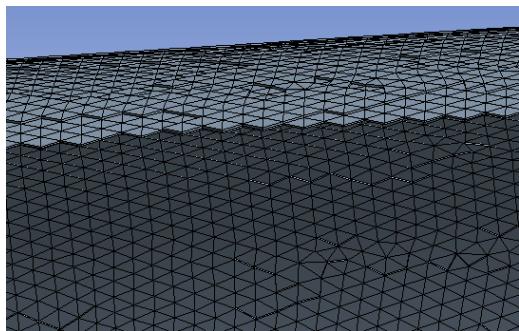


Illustration [26].

Aspect ratio of 17.8

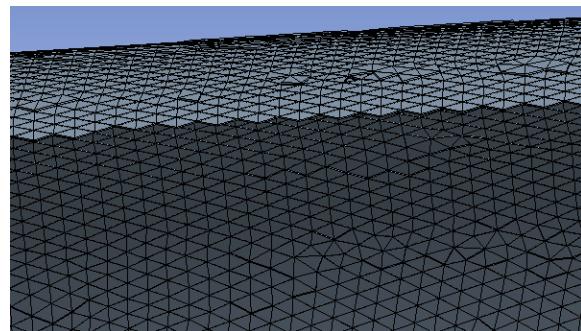


Illustration [27].

Aspect ratio of 22.5

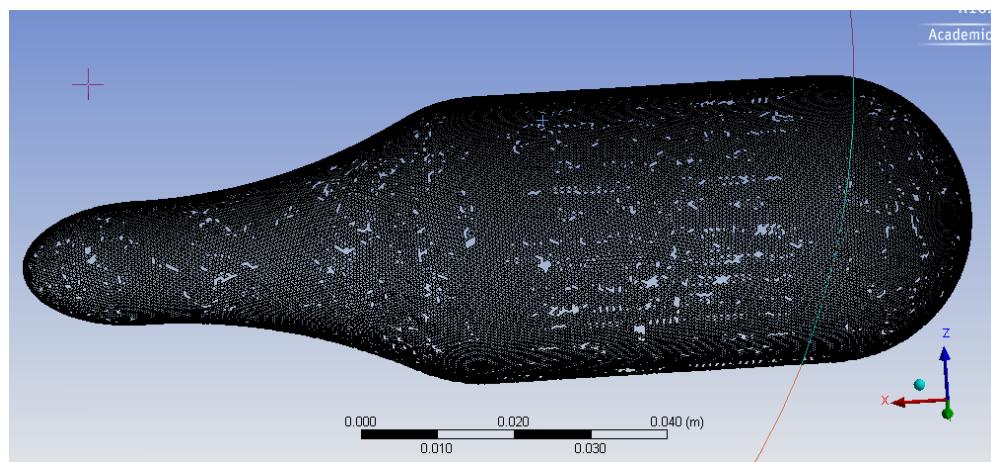


Illustration [28].

Aspect ratio of 36.8

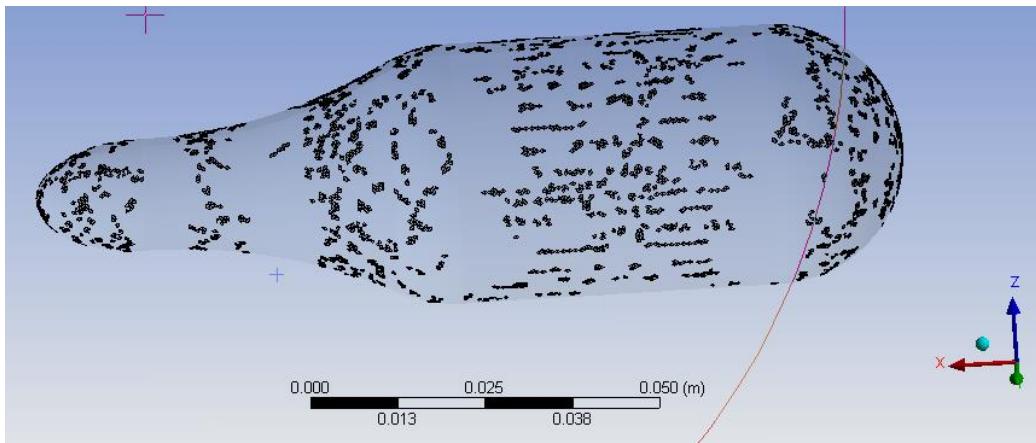


Illustration [29].

Aspect ratio of 41.5

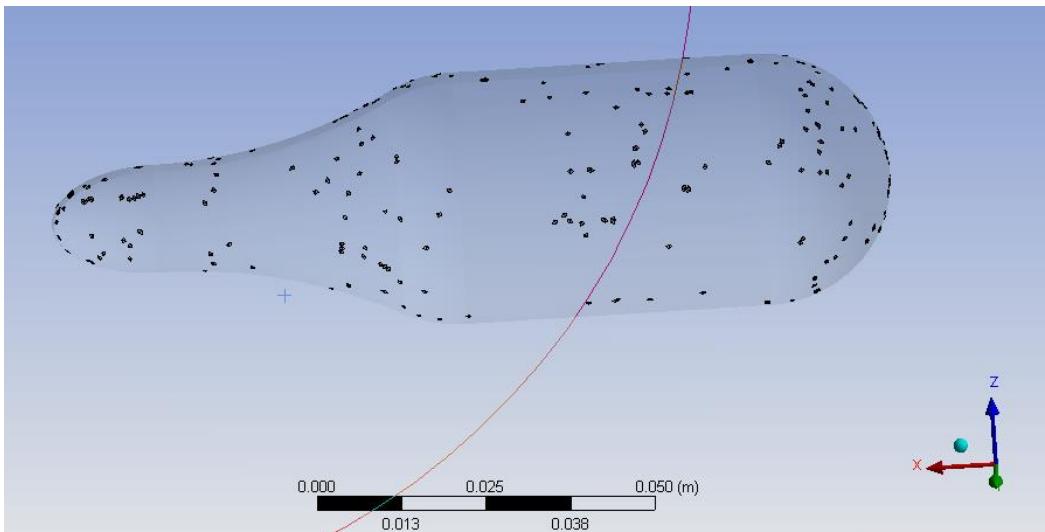
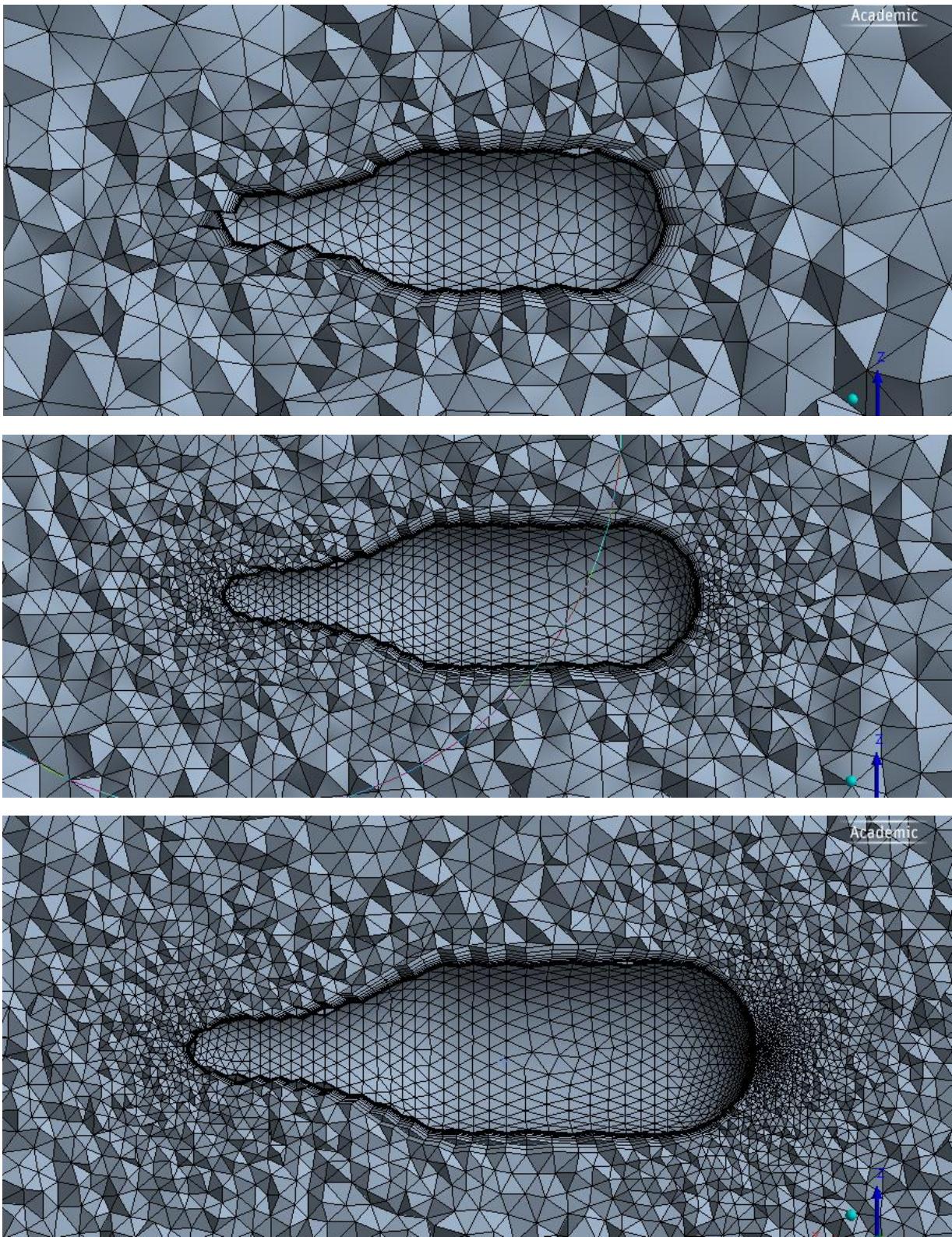
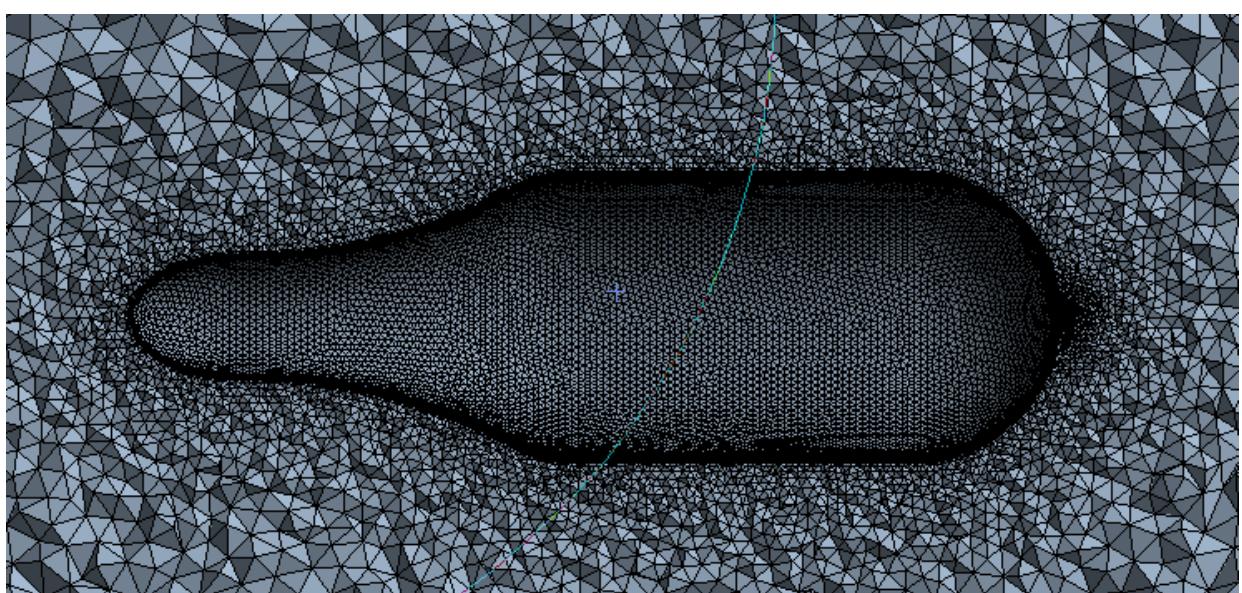
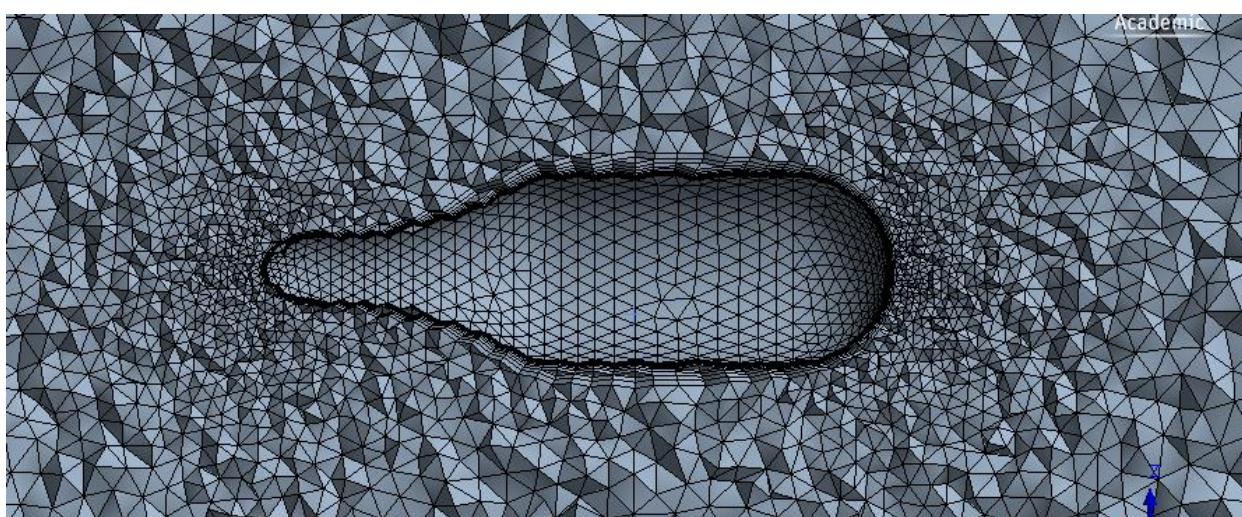
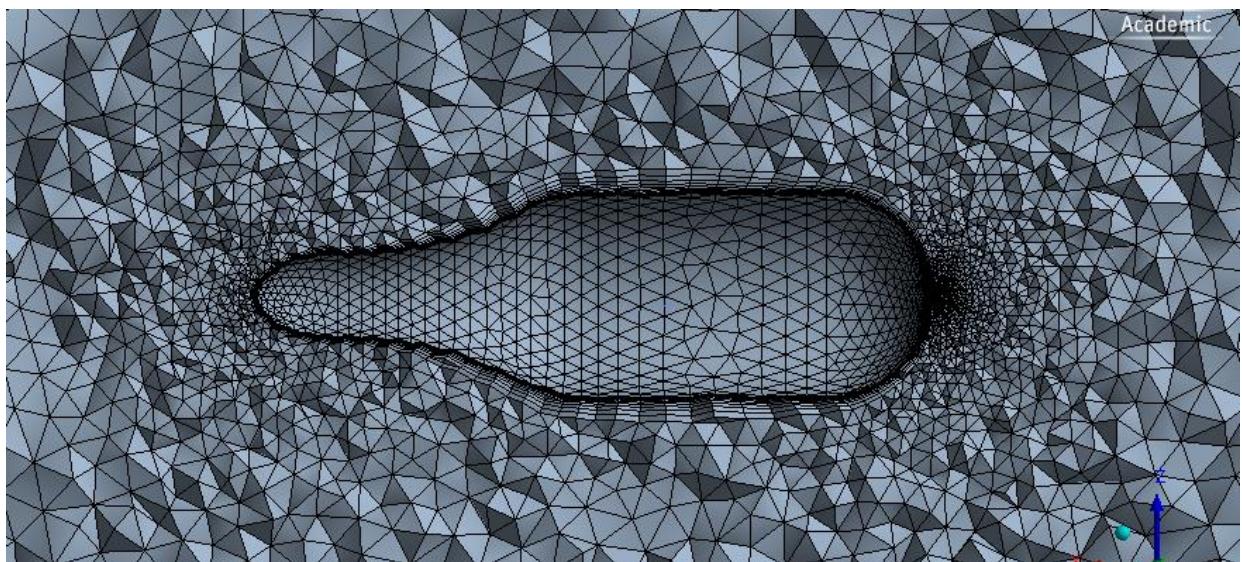


Illustration [30].

Comparison of the meshes.





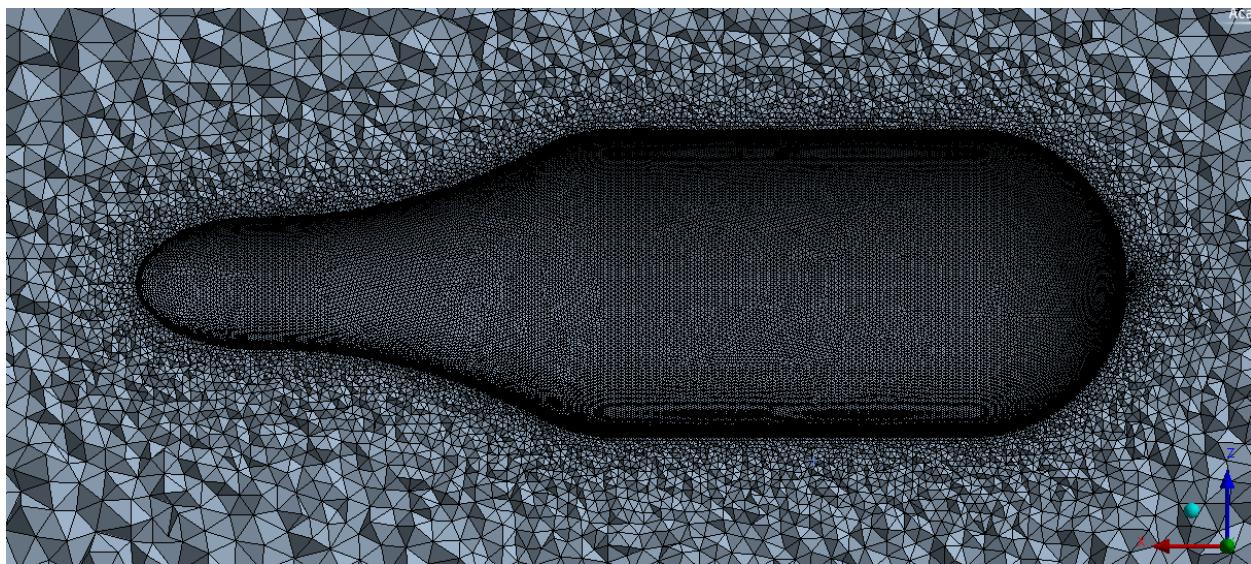
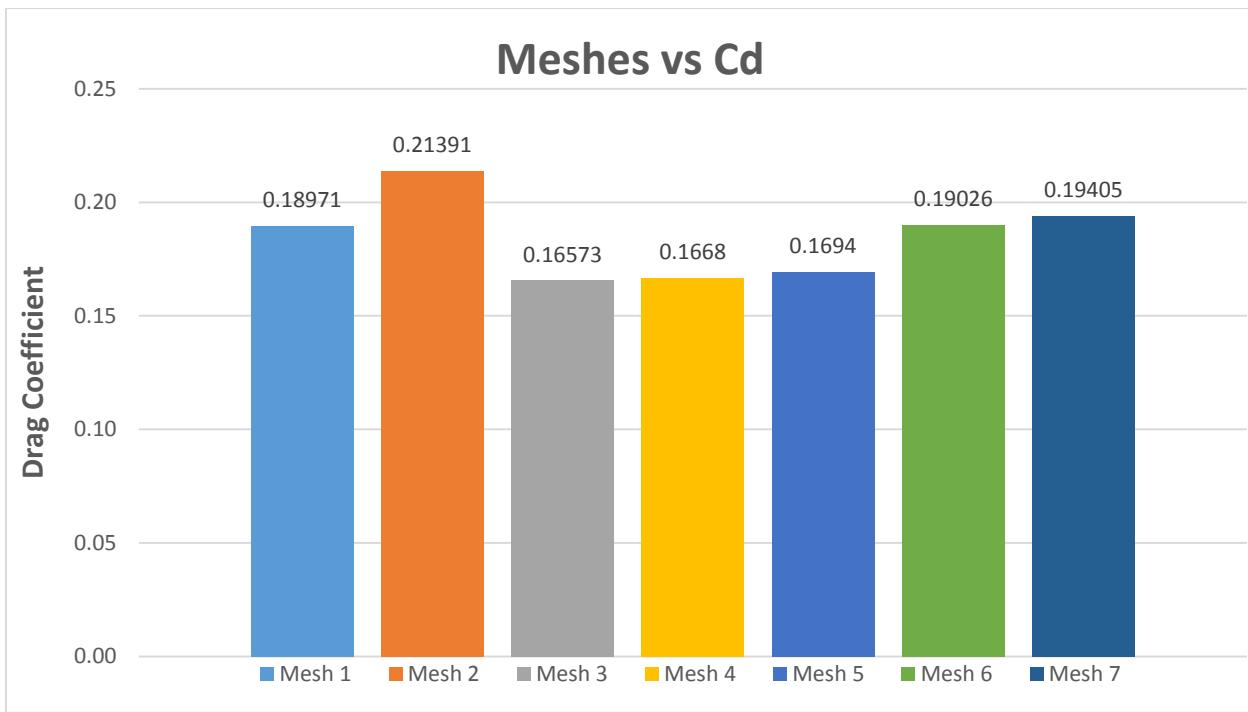


Illustration [31].

After consideration and meticulous study, we have decided to take the mesh number four as the optimum mesh of our study



Plot [6]

As, we can see in the graph number (6), we have obtained results very close to each other, nonetheless after mesh number 2 the results are even closer. So, the main reason why I have chosen the mesh number 4 is because the results of this mesh works in an order of magnitude good enough and the number of cells is accepted for our system.

As I have already said, it is difficult to predict a perfect mesh because it does not have a given reference patron to keep in track. So our decision is up to our experience with Fluent (Ansys) and how match the results respect similar basic geometries or different meshes made it with the same aim.

As a rule of thumb, a good method to ensure the simulation reliability is doing experiments, in this way we can match both results.

In the table below, it can be found the setting for the mesh number four and a briefly explanation about why this setting was chosen.

Settings	Selections	Justification
Solver Preference	Fluent	It has been selected Physics Preference CFD obviously, this mesh is used for CFD problems, with a relevance of 0.
Units	Metrics (m,Kg,N,s,V,A)	This units are due to the parameters used to calculate the drag coefficient that is why it considers [m] the most suitable unit. On the other hand, it is advisable the use of international units.
Method	Patch Conforming Method	Instead of setting none as an option, we have considered Tetrahedrons as method for the whole domain. That gives a more regular cells distribution in the fluid.
Sizing	Proximity and Curvature Proximity Size Function Sources Smoothing	It provides suitable minimum and maximum cells. The minimum size is 0.00004 m and the maximum 0.2 m. This command make us improve the distribution of cells along the shape. Medium.

Inflation	Wall- turbine	As global Inflation is available for 3D geometries such as this, we have done it. In this inflation it was used an inner and upper boundary. This command allows us to modify and improve the meshing at the wall, by this mean we solve the boundary layer properly.
Orthogonal Quality	0.87227	We can see here that our orthogonal quality is low, and our aspect ratio is also off (2.5752). However we have done 7 different meshes and we can tell that the results are not far from each other.
Nodes/Cells	137495/726918	It is considered an appropriate amount, which can solve the simulation properly in a suitable range of time.
Growth Rate	Sizing 1.13 Inflection 1.3	I select 1.13 because I do not consider very important the domains far from the airfoil. In the inflection I set up the growth rate to solve the boundary layer.
Named Selections	4 Named Selections	Velocity-inlet. Wall-turbine. Pressure-outlet. Symmetry.
Other parameters	Reynolds Number ~ $\Delta s \sim$ $Y^+ =$	2.5 e5 (Turbulent flow) 5 e-5 5 (Optimize to solve the boundary layers properly.)

Table [3].

Boundary layer.

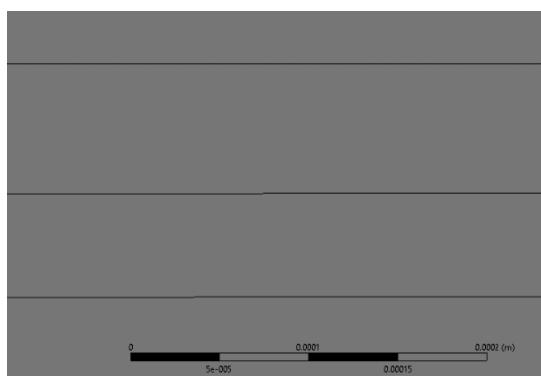


Illustration [32].

Input		Output	
<input type="button" value="Reset to Sea Level Conditions"/>			
U_∞ :	2	freestream velocity (m/s)	Δs :
ρ :	998.2	freestream density (kg/m ³)	0.000053483628713600 wall spacing (m)
μ :	0.001003	dynamic viscosity (kg/m s)	Re_x :
L :	0.124	reference length (m)	246813.1605184447 Reynolds number
y^+ :	5	desired y^+	Note: -1 indicates an input error

Illustration [33].

With the aim to clarify the Fluent model setting, below there is a briefly explanation about why this setting was chosen:

Settings	Selections	Justification
Fluent Launcher	Double Precision Processing Options	Parallel (Local Machine) Solver Processes 8 because the computer, which I am using for this project has a processor Intel inside XEON with 8 cores.
Solver Type	Density-Based	The density-based solver solves the governing equations of continuity, momentum, and energy and species transport simultaneously, this is how we have considered this problem.
Time Scheme	Steady	The rotatory device is going to be considered stopped and the fluid field is moving through the airfoil. Then, we do not have any need to study the transient case so far.
Models	Viscosity	Standard k-e. Robust. Widely used despite the known limitations of the model. Performs poorly for complex flows involving severe pressure gradient, separation, and strong streamline curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies.
Fluid Material	Liquid- Water	It is selected an incompressible flow. The Fluid material is take it from the Data-Base of fluent. The main properties of this is: Density of 998.2 Kg/m ³ Viscosity of 0.001003 kg/m-s
Airfoil Material	Aluminum	It is a good choice for this rotated device, because it is being immersed in water, avoiding corrosion and rusted issues. This is also a light material. Anyhow, the tests will be in plastic, but these make no significant differences.

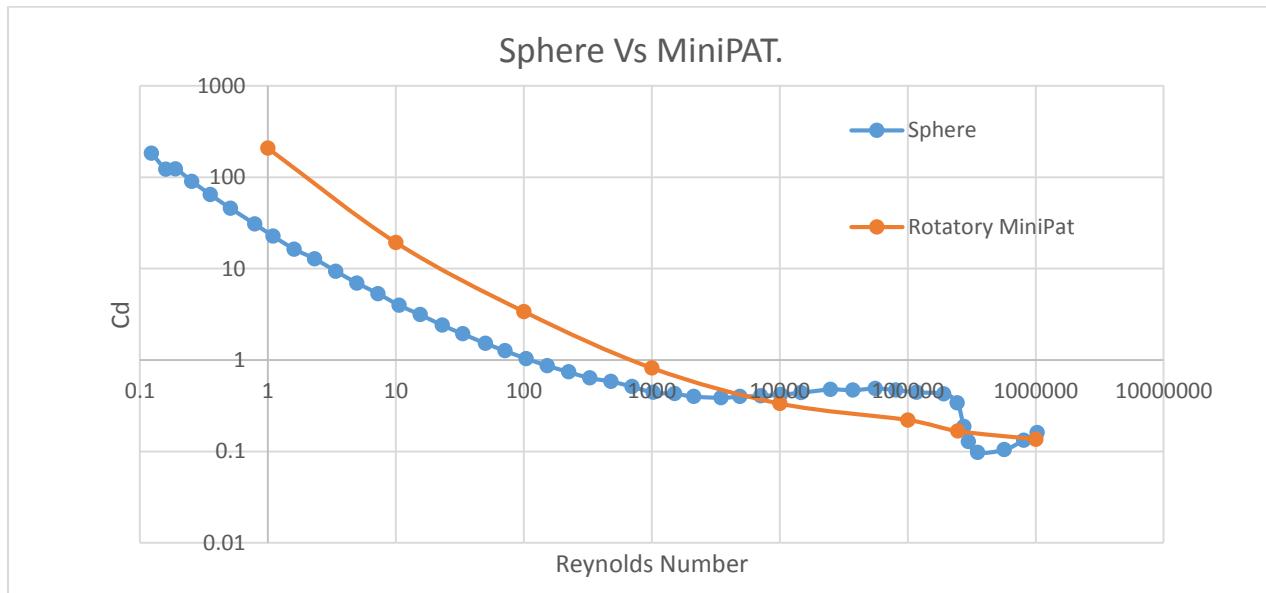
Zone Conditions	Turbine3d	This is set as a fluid, (water-liquid) with a fixed reference frame.
Boundary Conditions	Velocity-inlet. Interior-trubine3d. Pressure-outlet. Wall-turbine. Symmetry	-Velocity inlet 2 m/s. -Interior. -Outflow. -Wall. -Symmetry. No thermal setting. At the inlet and outlet: Turbulent viscosity ratio: 10 Turbulent Intensity: 5 %.
Reference Values	Compute from velocity- inlet. Area. Length.	We have the most significant date at this zone. <i>'Reference values for velocity, density, temperature, etc. will update from the free-stream values as described on the previous slide'</i> That means that the calculations are going to start from the Fairfield to the rest of the meshing. Cross section area: 0.00113 m^2 Chord: 0.124^2
Solution Scheme	*Solution Methods *Solution Controls *Solution Monitors *Solution Initialization	*Implicit formulation and Roe-flux type. This is more stable, we can have results faster than explicit method. Least Squares Cell Based is chosen because it does not have any issue with turbulent viscosity ratio. Second Order Upwind for flow and turbulence discretization. That provides a better accurate drag in comparison with the first order schemes, which is not sufficient. *The Courant number determines the internal time step used by the density based solver and therefore it affects the solution speed and stability. The default Courant number for the density-based implicit formulation is 5.0, but as we are using automatic 'solution steering'. * We must set up the draft and lift plots and also print the results. In this part we are going to introduce the angle of attack information by the force component (in the case we want to have a different angle of attack than 0). *Let's initialize the simulation from the velocity-inlet as we said. This selection has all the parameters already set up.

	*Solution Steering	*I have used 'Full-Multi-Grid (FMG) Initialization' which will compute a quick, simplified solution based on a number of coarse sub-grids. This quick solution can help to get a stable starting point and is a better 'initial guess' for the main calculation. The flow type is incompressible because of the Mach Number. Explicit Under Relaxation Factor equal 0.75. The reporting interval is 20, which make the calculation faster.
Monitors	Residuals, cd & cf.	I have plotted residuals, cd and cf. These are going to compare with experimental data. In the plots and prints we can see how the results converge.
Initialization Type	Standard Initialization	This command make a simpler initialization starting from the far field.
Convergence Criteria	None	It has no convergence criteria, we can see if the convergence is appropriate or not from our own knowledge.

Table [4].

Once we have picked the best 3-D mesh, we are going to plot a graph with drag coefficient as a function of Reynolds number for a sphere and rotatory MiniPAT.

Here it can be seen, how both plots are represented with similar behavior. The slope of these curves are quite similar long to the Reynolds number, however we can appreciate how the slope of sphere has a dramatic decline between 2×10^5 and 6×10^5 . This decline does not show up in our simulation results.



Plot [7]

Simulation of the drag with an angle of attack of 10°.

Let's calculate the drag with an angle of attack of 10° because marine animals will pull the device not perfectly straight.

To calculate the drag coefficient, it needs to be calculate the cross section area. For this case this area is 0.00115 instead of 0.00113.

This area can easily be calculated in SolidWorks.

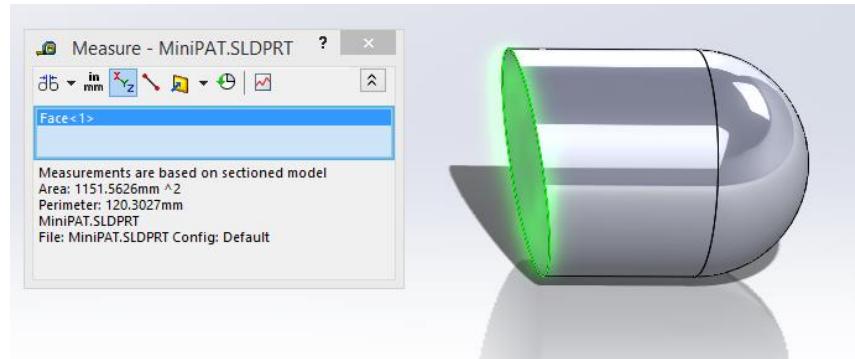


Illustration [34].

To set the angle of attack in fluent, we must introduce the factors components in the Force Vector case, thus it is totally defined an angle of attack of 10°.

The drag coefficient obtained is seen in the illustration below:

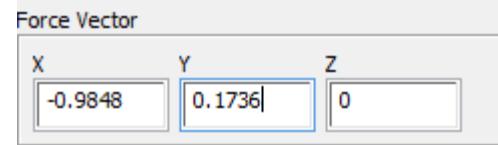
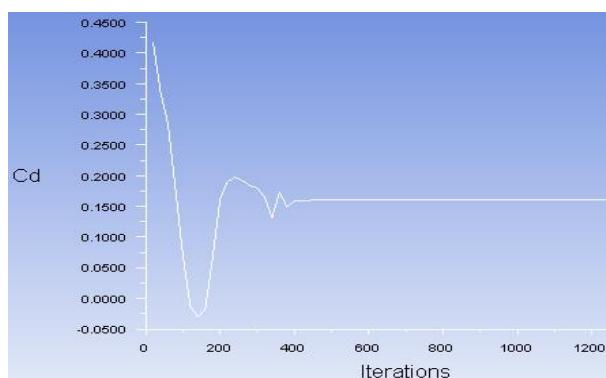


Illustration [35].



The difference between null angle of attack and 10° is neglected.

iter	Cd-1	time/iter
1120	1.6093e-01	0:37:08 880
1140	1.6093e-01	0:36:12 860
1160	1.6093e-01	0:35:25 840
1180	1.6093e-01	0:34:30 820

Illustration [36]

Three Dimensional simulation. Fluted

With the aim to accomplish the requirements of this device. It has been designed two final model and it has been chosen one of them. I want to inform to the reader that, this final device is just a prototype, what we are going to focus on. The goal of this project is to find the optimal shape which produce the maximum torque with the less drag force.

Now, next part of this section is going to talk about the simulation of these two models and we will able to appreciate, the reasons why we pick one of the models or the other one.

In the illustrations below, we find the two different geometries.

Model 1

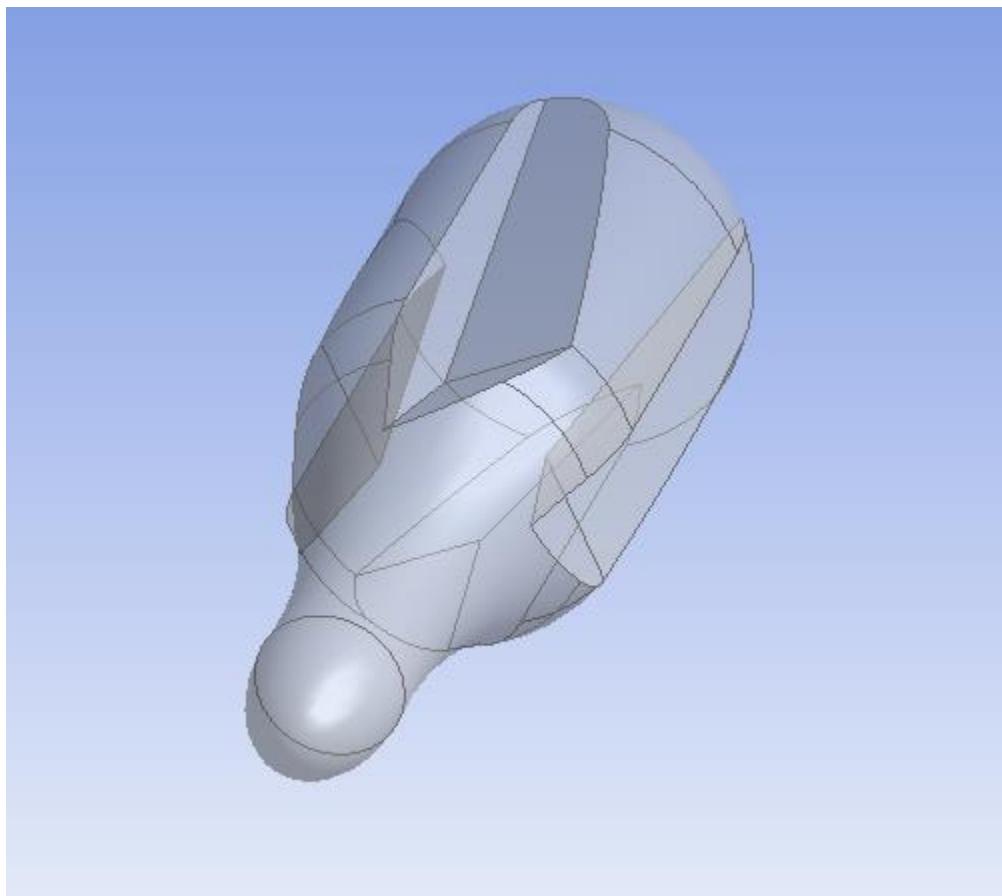


Illustration [37]

In order to identify which model gets maximum performance, it has been printed a couple of illustration. By these imagines we can figure out which model develops higher performance by visual inspection.

Path lines.

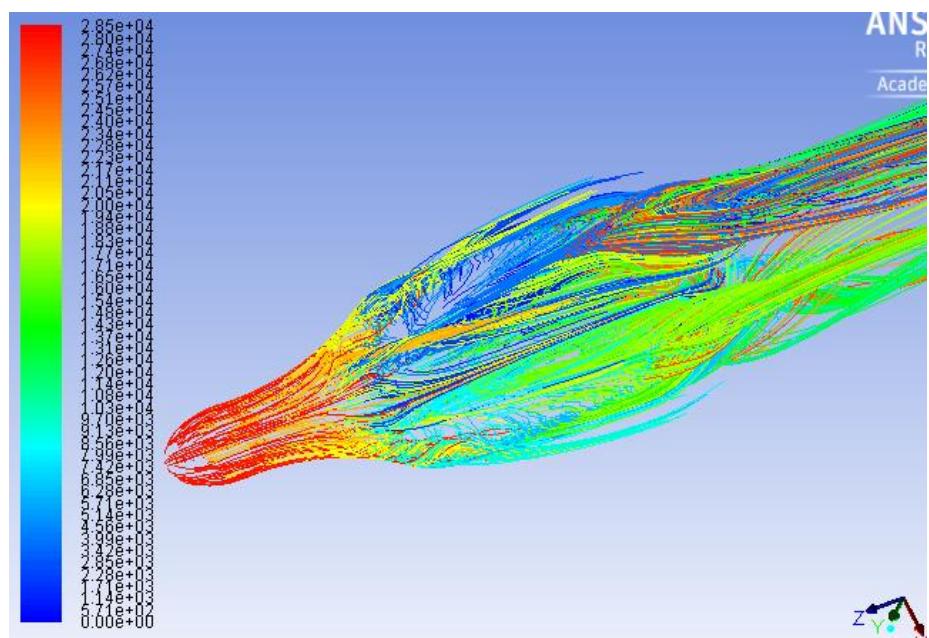


Illustration [38]

Vector velocity.

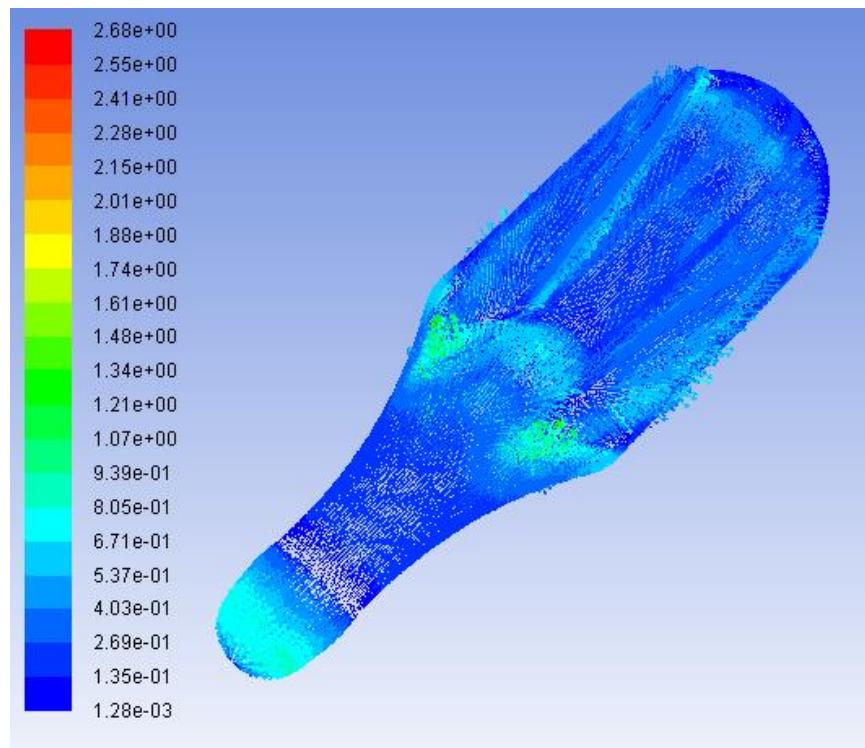


Illustration [39]

Pressure contours

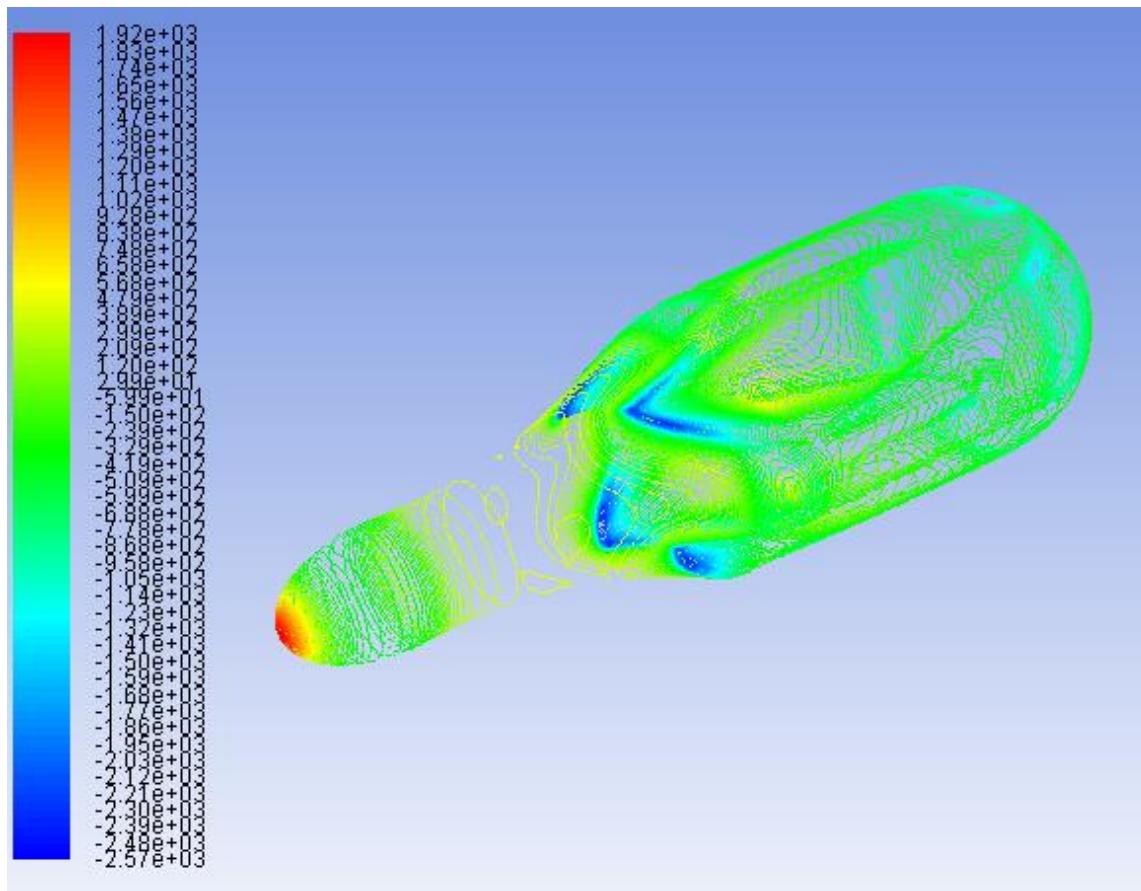


Illustration [40]

Once, it has visually analyzed, we can appreciate some singularities at the incoming of the fluted part (dark blue in the illustration [40]).

Let's focus on the path lines plot and pressure contours. We can see at the fluted inlet how it shows a low pressure region, which produce a turbulent region absolutely unnecessary.

We can easily predict that this turbulent regimen produce an awful drag force. For this study we are avoiding drag forces with the aim to reduce drag coefficient the most.

In the illustration [41] we can see an upgraded model, where the low pressure are suppressed at the inlet flutes. This model is improved however it still has some drawbacks which are going to take into account later.

Model 2

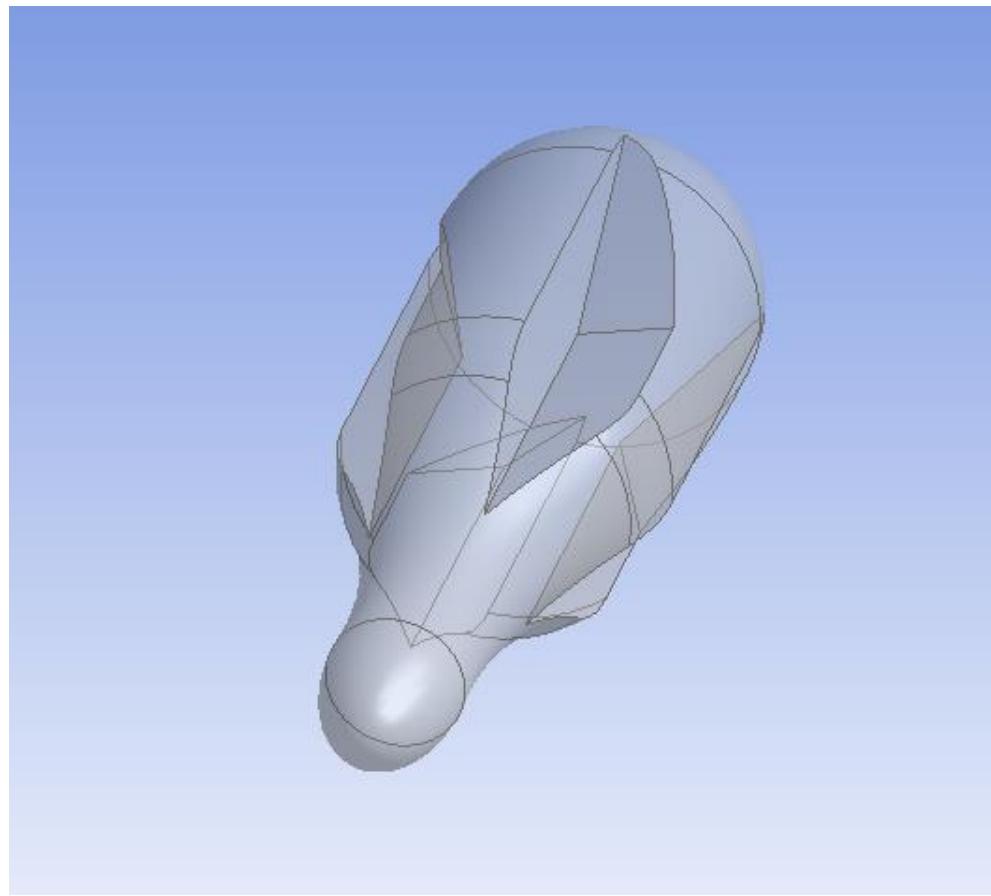


Illustration [41]

In the illustration below, we can directly observe, how the pressure contours are regular at the rotatory MiniPat. In this model the flow through the flutes inlet is smoothly till the flow comes out from the MiniPat producing some vortexes which are going to make the device to rotate.

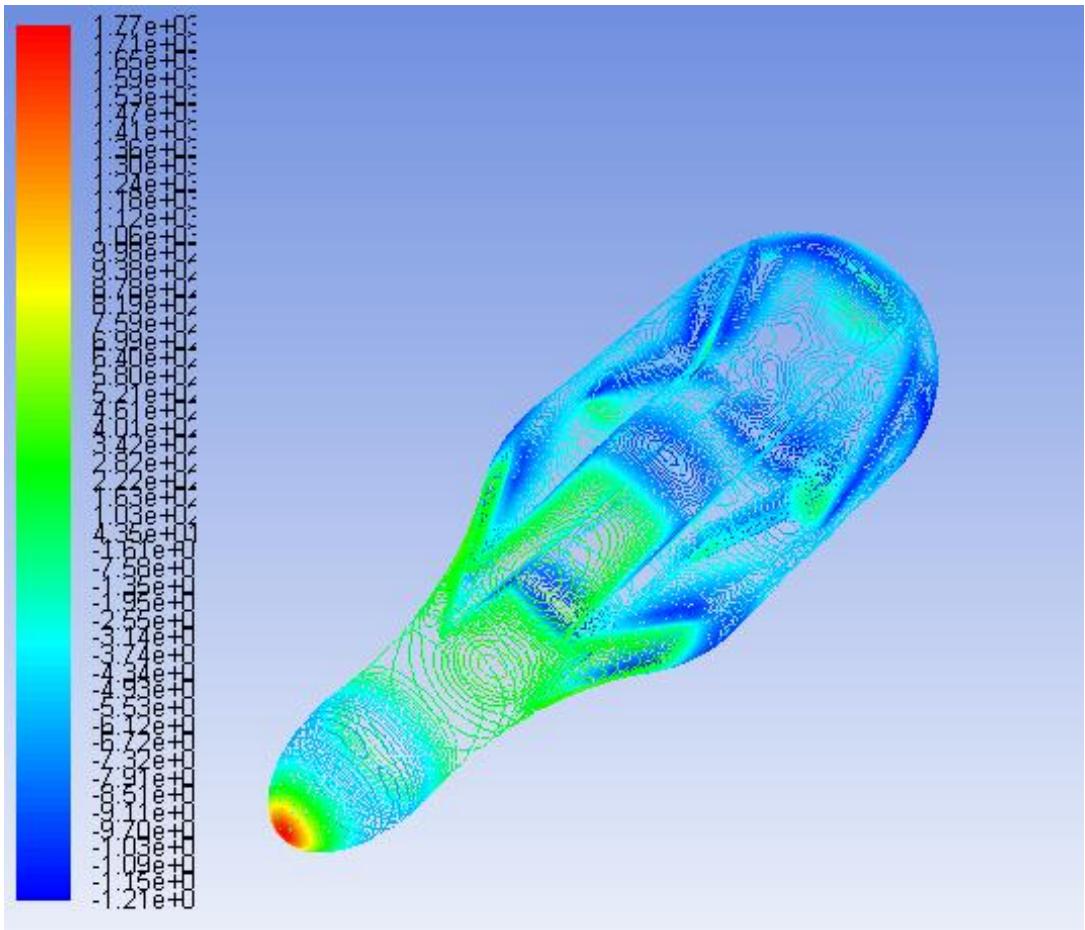


Illustration [42]

MESH

For this mesh, we have used the same techniques as the non-fluted model. These techniques do a good job at the most complex parts of the geometry as well as edges. The boundary layer are properly resolved by the inflation layers and Standard Wall Functions.

The following illustrations show the main features of this mesh.

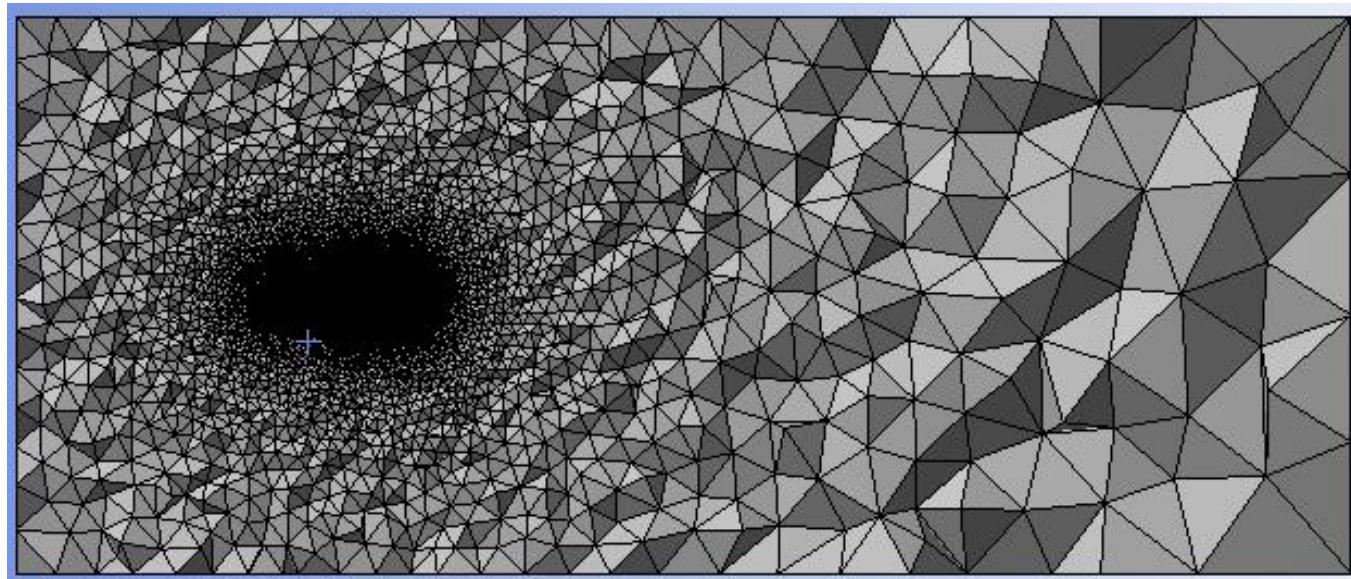


Illustration [43]

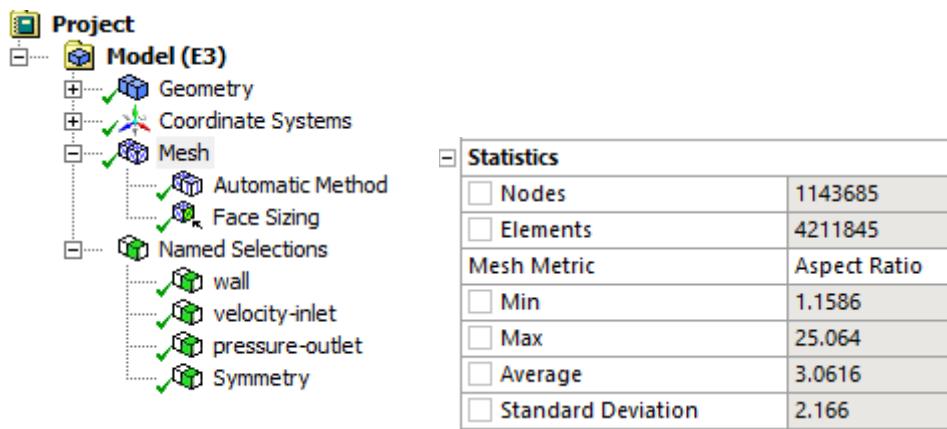


Illustration [43]

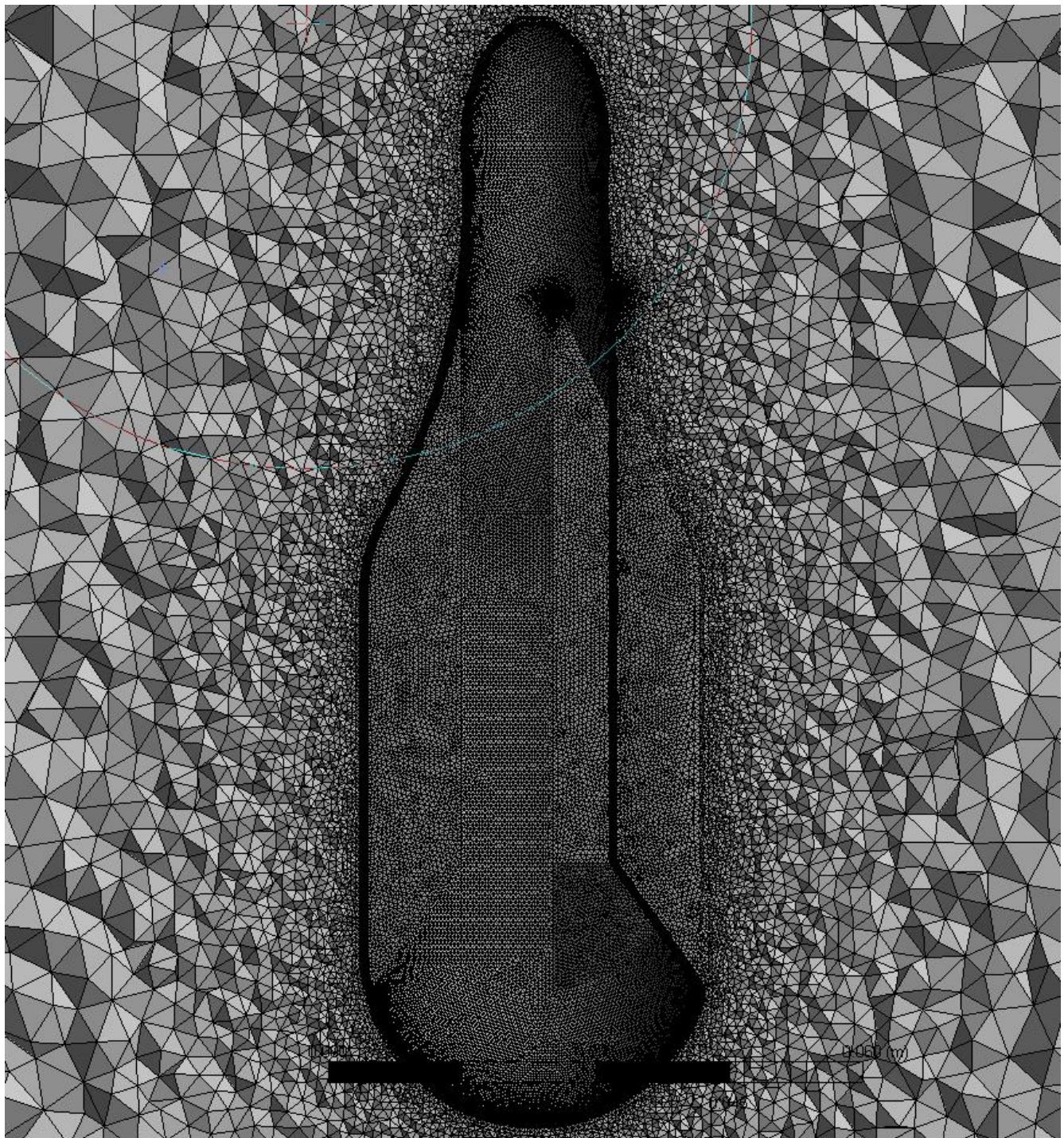


Illustration [44]

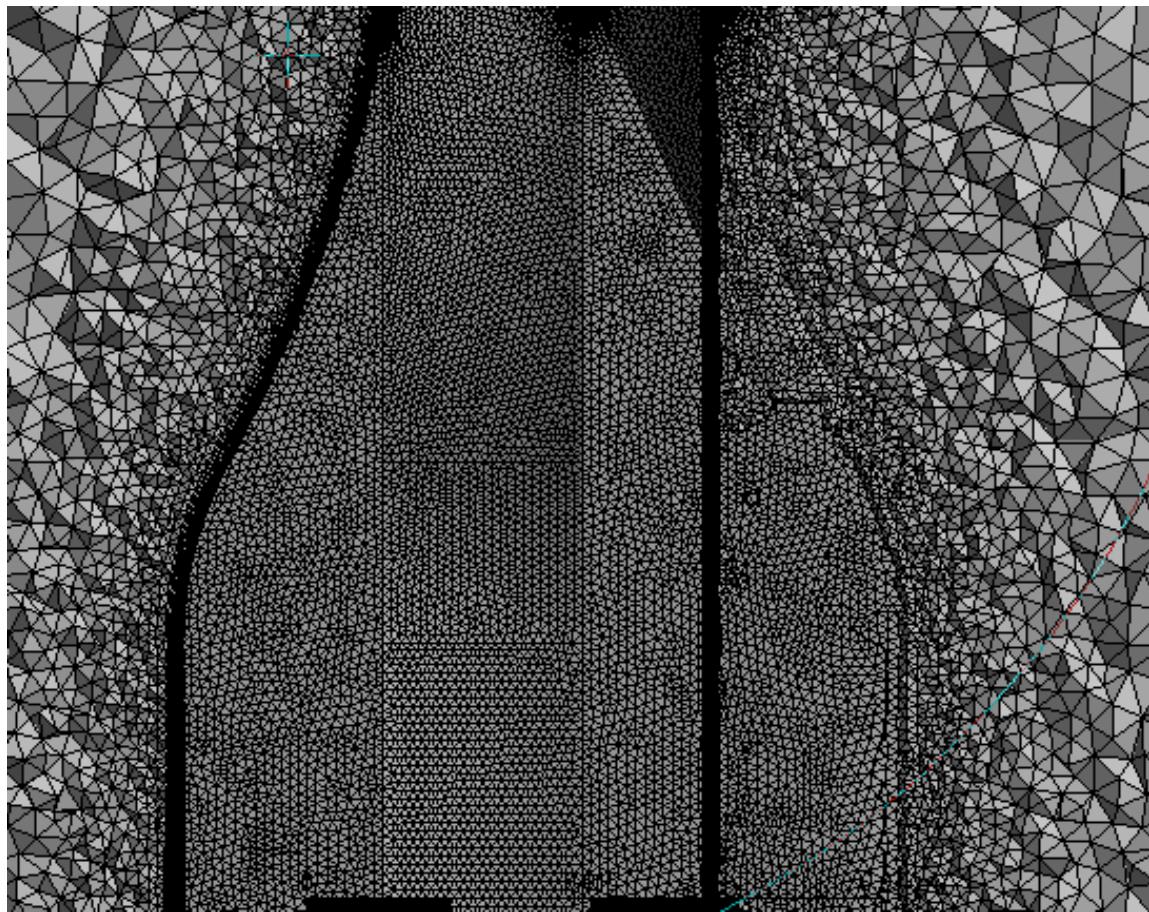


Illustration [45]

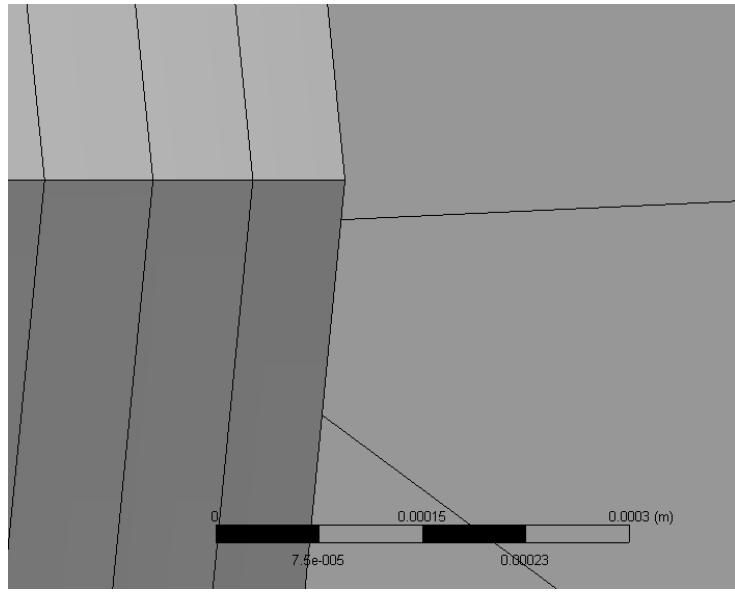


Illustration [46]

Aspect ratio.

As we already know, aspect ratio is a parameter which let us know about the quality of the mesh. In our mesh we can find a maximum and minimum of these parameter. The maximum aspect ratio appears at the wall because it is also set an inflation with the aim to solve the boundary layers. In the illustration below it can be observed the cells with an aspect ratio of 12, we can see there are no too many of them. I defiantly classify this mesh as a high quality mesh certainly heavy.

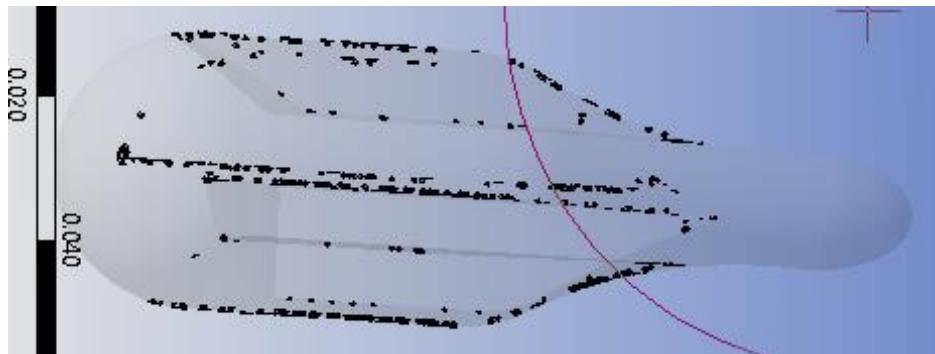


Illustration [47]

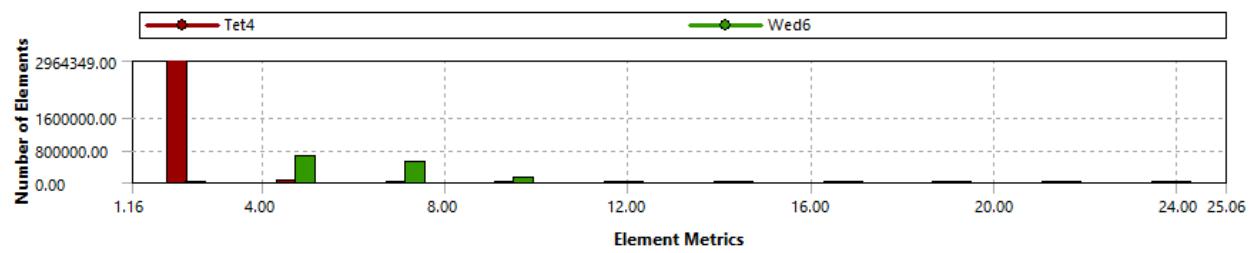


Illustration [48]

Torque.

In order to calculate the torque of this rotatory MiniPat, we are going to calculate the forces at one of the four surface which produces torque respect to the devices axis. In the next picture we can see how the model for this simulation looks like, the surface is highlight.

The calculated total force in this surface in the plane x-y, ignoring the coordinate z because this coordinate is in the same devices axis direction.

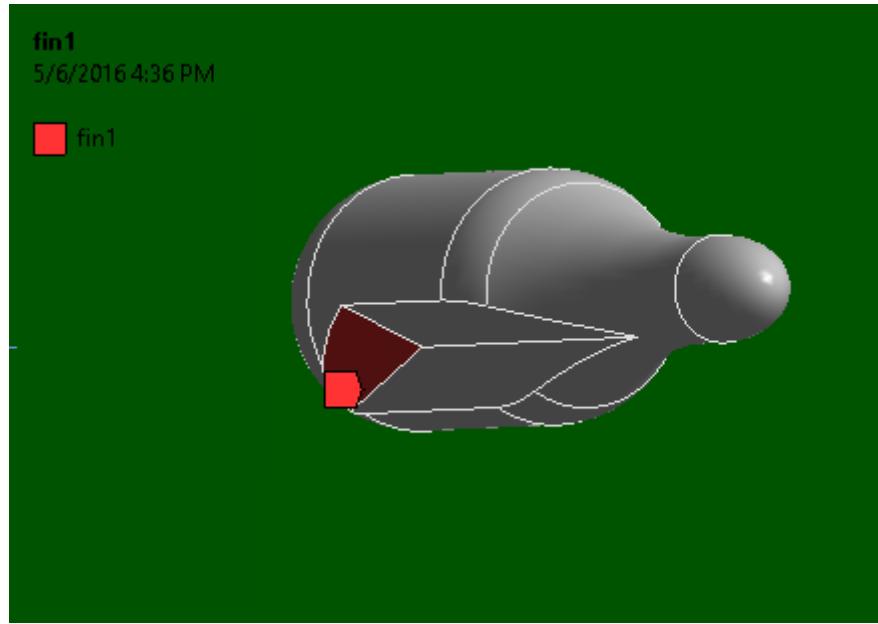


Illustration [49]

For this simulation we have obtain:

Forces - Direction Vector (0 1 0)			
	Forces (n)		
Zone	Pressure	Viscous	Total
fin1	2.1829742e-16	0.00010149549	0.00010149549
<hr/>			
Net	2.1829742e-16	0.00010149549	0.00010149549

Forces - Direction Vector (1 0 0)			
	Forces (n)		
Zone	Pressure	Viscous	Total
fin1	0.076406631	-0.00060472632	0.075801905
<hr/>			
Net	0.076406631	-0.00060472632	0.075801905

Illustration [50]

The force acting at the fin surface in the y-direction is negligible.

The final torque is the x-direction force multiply by the radius (as an approximation) times four flutes. See results at the Results section of this report.

Experiments.

In this section, it is going to be presented the experimental methods of this project. This part of the project is going to be vital to match our results from Fluent and ensure that our simulation is right as well.

To achieve this goal, we need to know the velocity of the fluid through the shape in the wind tunnel. For this, we are using dimensional analysis method, by the Reynolds number we can find a proper velocity in the wind tunnel. Nevertheless, before calculating the Reynolds number, we need to calculate some parameters in the wind tunnel because we are running our experiments in Flagstaff, son we need to calculate the density, dynamic viscosity and corrected velocity in Flagstaff.

Density:

Let's solve for the density:

By the barometric pressure in flagstaff:

$$P = P_o \left(1 - \frac{L h}{T_o}\right)^{\frac{g M}{R L}} = 101325 \left(1 - \frac{0.0065 \times 2106}{288.15}\right)^{\frac{9.81 \times 0.02896}{8.314 \times 0.0065}} = 78452 \text{ Pa}$$

P = Pressure at h altitud, [Pa]

P_o = Sea level standard atmospheric pressure, [Pa]

L = Temperature lapse rate for dry air, [$\frac{K}{m}$]

h = altitud, [m]

T_o = Sea level standard temperature, [K]

g = Earth – surface gravitational acceleration [$\frac{m}{s^2}$]

M = Molar mass of dry air [$\frac{kg}{mol}$]

R = universal gas constant $\left[\frac{J}{mol K}\right]$

ρ = density [$\frac{kg}{m^3}$]]

$$\rho = \frac{P}{R T} = \frac{78452}{287.058 \times (23 + 273)} = 0.923 \frac{kg}{m^3}$$

Velocity Calibration (Wind Tunnel)

The velocity which is shown at the wind tunnel software is dis-calibrated due of the wrong density (Sea level density). We have to recalibrate the velocity using the corrected density in flagstaff. With the aim to achieve this goal, I have defined a calibration function.

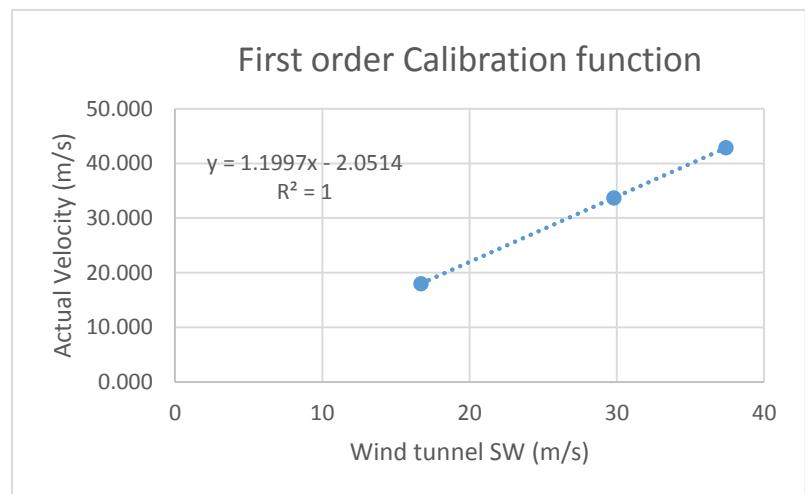
I have taken different velocities from the Pitot tube installed right next to the specimen. These velocities are right. I have written down the software velocities. Finally the function that it can be obtained in excel is:

By taking the total and static pressure from the Pitot tube, we can resolve for the actual velocity going through the wind tunnel:

$$V = \sqrt{2 \times \left(\frac{P_{total} - P_{static}}{\rho_{Flagstaff}} \right)}$$

Density			PITOT TUBE			
0.923	Actual Velocity (m/s)	Wind tunnel SW (m/s)	Total pressure (" H ₂ O)	Static pressure (" H ₂ O)	Total pressure (Pa)	Static pressure (Pa)
	17.995	16.7	1.7	1.1	423.436	273.988
	33.666	29.8	3.2	1.1	797.056	273.988
	42.837	37.4	4.55	1.15	1133.314	286.442
	CALIBRATION F.					
Ex.	37.09	32.62598983				

Table [6].



Plot [8]

So, finally if we want an actual velocity of 37.09 m/s, we have to type in the program the amount of 32.63 m/s. That is the result obtained by getting 'x' value from the calibration function.

Forcrometer Calibration (Wind Tunnel).

With the aim to make sure that we get the right results, we need to calibrate de forcemeter or make sure that the results are having the given error, how the company inform us.

For this, we have use test mass of 20, 50 and 100 grams which have a weight of 0.20, 0.49 and 0.98 respectively. To proceed with this calibration we are going to place the mass vertical respect the axis of the rod and measure the drag force which appears in the AeroLab Software. Here we can see the results:

Illustration [51]



Force Meter Calibration		
Mass (g)	Actual Force (N)	Recorded Force (N)
20	0.20	0.21
50	0.49	0.49
100	0.98	0.98

Table [6].

After taking a look at the results we can see that they match quite well. We have some error for low drag forces, however for high speed the results are very accurate. Anyhow the results agree with the accuracy of the forcemeter which is ± 0.01 N

We are going to test a 1.5/1 scale device, because we want to get high drag forces which allow us to avoid error, on the other hand it must be said that the error for low speed is not that alarming.

Dimensional Analysis:

The next step in this project is to find the right parameters with the aim of launching experiment in the wind tunnel. To reach this goal, it will take into consideration 'Buckingham Pi Theorem'.

The interest of this project is to solve the drag coefficient in a marine turbine geometry. So, the main parameters, which drag F depends on, are the chord C , fluid density ρ , viscosity μ and fluid speed V .

Some parameter as ϵ , size of roughness and a , sound speed are negligible because they are unconsidered meaningful in the experiment.

After this considerations it obtains:

$$F = F(C, \rho, \mu, V)$$

The nature of function:

$$g(F, C, \rho, \mu, V) = 0$$

Following the Buckingham pi theorem, the fundamental dimensions are ($K = 3$):

m = dimensions of mass

l = dimensions of length

t = dimensions of time

The physical variables and their dimensions are ($N = 3$):

$$\left\{ \begin{array}{l} [F] = m l t^{-2} \\ [\rho] = m l^{-3} \\ [V] = l t^{-1} \\ [C] = l \\ [\mu] = m l^{-1} t^{-1} \end{array} \right.$$

The dimensionless Π products are: $N - K = 2$

$$\begin{aligned} f_2 &= (\Pi_1, \Pi_2) \\ \Pi_1 &= f_3(\rho, V, C, F) \\ \Pi_2 &= f_4(\rho, V, C, \mu) \end{aligned}$$

For the time being, concentrate on Π_1 . Assume that:

$\Pi_1 = \rho^d V^b C^e F$, where d, b and e are exponents to be found. In dimensional terms.

$$[\Pi_1] = (m l^{-3})^d (l t^{-1})^b (l)^e (m l t^{-2})$$

$$m \rightarrow d + 1 = 0; \quad d = -1$$

$$l \rightarrow -3d + b + e + 1 = 0; \quad e = -2$$

$$t \rightarrow -b - 2 = 0; \quad b = -2$$

$$\Pi_1 = \rho^{-1} V^{-2} C^{-2} F$$

$$\Pi_1 = \frac{F}{\rho V^2 C^2} \rightarrow \frac{D}{\frac{1}{2} \rho V^2 A} \text{ Drag Coefficient}$$

It evaluates $\Pi_2 = \rho V^h C^i \mu^j$, where h, i and j are exponents to be found. In dimensional terms.

$$[\Pi_2] = (m l^{-3}) (l t^{-1})^h (l)^i (m l^{-1} t^{-1})^j$$

$$m \rightarrow 1 + j = 0; \quad j = -1$$

$$l \rightarrow -3 + h + i - j = 0; \quad i = 1$$

$$t \rightarrow -h - j = 0; \quad h = 1$$

$$\Pi_2 = \rho V^1 C^1 \mu^{-1}$$

$$\Pi_2 = \frac{\rho V C}{\mu} \text{ Reynolds Number}$$

By this dimensional analysis it can be written:

$$f_2 = \left(\frac{D}{\frac{1}{2} \rho V^2 A}, \frac{\rho V C}{\mu} \right) \rightarrow f_2 = (C_d, Re)$$

$$C_d = f(Re)$$

One more regard about this equation is the influence of one more parameter α , angle of attack:

$$C_d = f(Re, \alpha)$$

Flow Similarity:

Consider two different flow fields over two different bodies. By definition, different flows are dynamically similar if:

1. The streamline patterns are geometrically similar.
2. The distributions of $\frac{V}{V_\infty}, \frac{p}{p_\infty}, \frac{T}{T_\infty}$, etc. throughout the flow field are the same when plotted against common non-dimensional coordinates.
3. The forces coefficients are the same. (This statement is consequence of the statement 2).

It can also say, two flows will be dynamically similar if the similarity parameters are the same for both flows. It has emphasized Re as parameter. For many aerodynamic applications, this is by far dominant similarity parameter. A flow over geometrically similar bodies at the same Reynolds numbers are dynamically similar, and hence the lift, drag and moment coefficients will be identical for the bodies.

After outlining this point, let it apply in this experiment for the Reynolds Number:

$$Re_{H_2O} = Re_{Air} \rightarrow Re_{H_2O} = \frac{\rho_{Air} V_{Air} C_{Air}}{\mu_{Air}} \quad \text{From Fluent's simulation it is known.}$$

$$Re_{H_2O} = 246813, \rho_{Air} = 0.923 \frac{kg}{m^3}, \mu_{Air} = 1.72 \times 10^{-5} \frac{kg}{m s}$$

$$246813 = \frac{0.923 V_{Air} C_{Air}}{1.72 \times 10^{-5}} = 53,662.8 V_{Air} C_{Air}$$

It has a relation between the two Reynolds numbers of these two different fluids. Now, it is going to obtain the suit dimensions and speed for testing in the wind tunnel. Before finding these two parameters it has to take into account the wind tunnel's dimension and airspeed range:

Test Section Dimensions – 12"x12"x24" (30.5cm x 30.5cm x 61cm)

Airspeed Range – 10 mph (4.5 m/s) to 145+ mph (65 + m/s)

Finally it is offered three different dimensions with three different speeds:

DIMENSION N° 1. $C_{Air} = 0.124 \text{ m}$	Speed N° 1 $V_1 = 37.09 \text{ m/s}$
DIMENSION N° 2. $\frac{C_{Air}}{2} = 0.062 \text{ m}$	Speed N° 2 $V_2 = 74.18 \text{ m/s}$
DIMENSION N° 3. $\frac{C_{Air}}{1.5} = 0.08267 \text{ m}$	Speed N° 3 $V_3 = 55.64 \text{ m/s}$
DIMENSION N° 4. $C_{Air} \times 1.5 = 0.186 \text{ m}$	Speed N° 4 $V_4 = 24.73 \text{ m/s}$

Table [7].

The highlighted dimension has been chosen because it is the only dimension which can reach an enough high drag force to be read by the forcemeter.

We must be aware of using the calibration function for this velocity, so if the actual velocity is 24.73 m/s the velocity which we have to set in the wind tunnel program is 22.5 m/s.

$$\text{Actual Velocity} = 24.73 \text{ m/s}$$

$$\text{Wind Tunnel Velocity} = 22.5 \text{ m/s}$$

Rapid Prototype.

In order to get the shapes and test them, we have decided to 3D print them. The plans of this devices will be found in the annex.

Non-Fluted MiniPat (Scale 1/1):



Illustration [52]

Non-Fluted MiniPat (Scale 1.5/1):



Illustration [53]

Fluted MiniPat (Scale 1/1):



Illustration [54]

Fluted MiniPat (Scale 1.5/1):



Illustration [55]

These devices have quickly been sanded, anyhow we have not considered viscous forces because the Reynolds number of our experiment is high enough to take it into consideration.

In the following images, we can see how the configuration looks like in the wind tunnel:



Illustration [56]



Illustration [57]



Illustration [58]

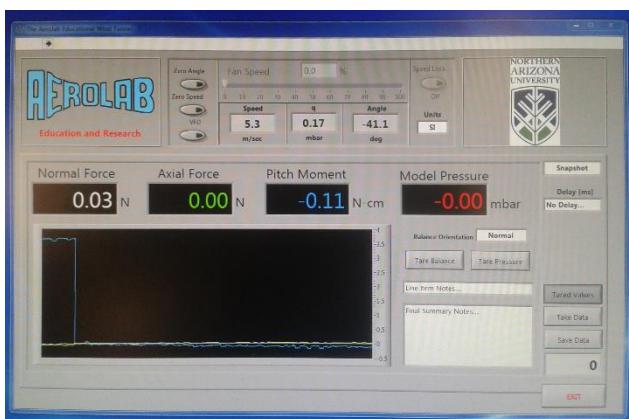


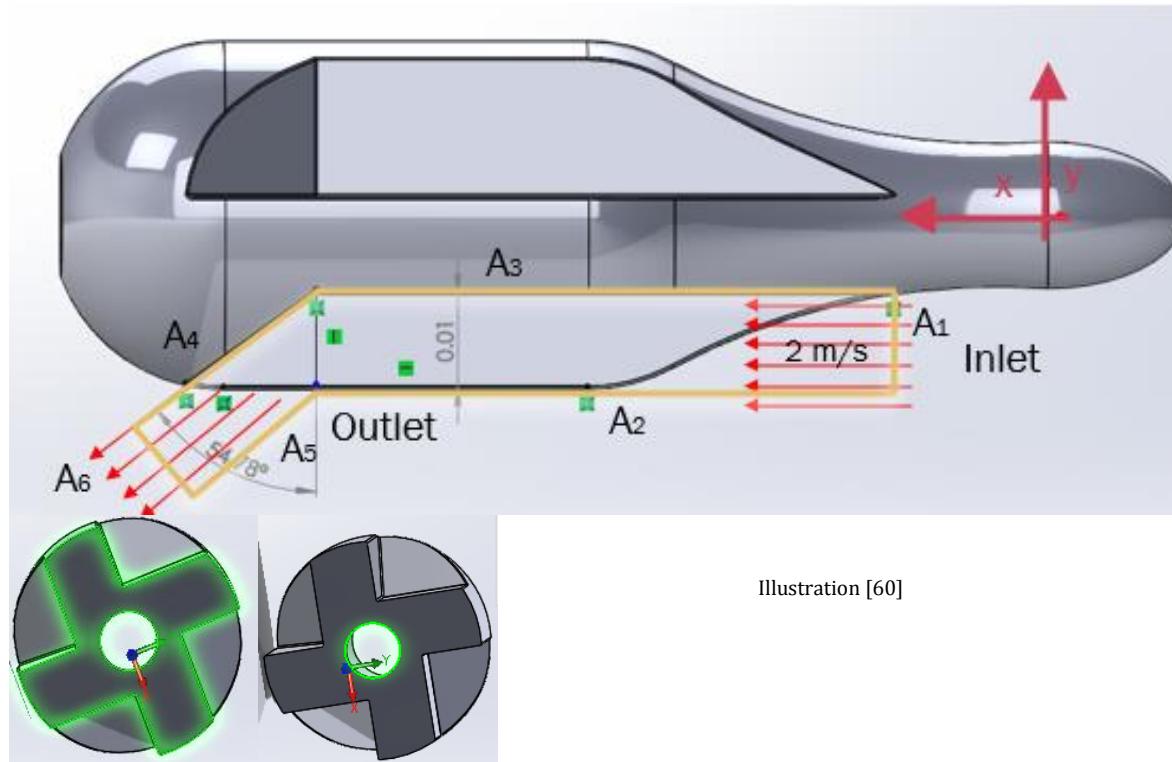
Illustration [59]

Theoretical Calculations.

In this section we want to calculate the torque by the fluid conservation equation.

Assumptions:

- Incompressible flow. (Water)
- Uniform flow at each section.
- $\bar{\omega} = \text{constant}$.
- Steady flow
- $A_1 = A_6$ and $A_2, A_3, A_4, A_5 \rightarrow v = 0, \text{no slip condition}$.



The two images under the problem sketch are used to get the areas in SolidWorks with the aim to find the inlet area A_1 .

Calculation of A_1 :

$$A_1 = \frac{A_{\text{total}} - (A_1 + A_2)}{N^{\circ} \text{ of flutes}} = \frac{(19^2 \pi) \text{mm}^2 - (625.28 \text{mm}^2 + (4.763 \text{ mm})^2 \pi)}{4} = 109.37 \text{ mm}^2$$

-Conservation of mass.

$$\frac{\partial}{\partial t} \int_{CV} \rho dV + \int_{CS} \rho \vec{V} d\vec{S} = 0$$

$\frac{\partial}{\partial t} \int_{CV} \rho dV$ is equal zero because is a steady flow.

$$\int_{CS} \rho \vec{V} d\vec{S} = 0 \rightarrow \int_{A_1} \vec{V} d\vec{S} - \int_{A_6} \vec{V} d\vec{S} = 0 \rightarrow \vec{V}_1 \times A_1 = \vec{V}_6 \times A_6$$

$$\vec{V}_1 = V_6 \sin 54.78^\circ \vec{i} + V_6 \cos 54.78^\circ \vec{j}$$

$$\vec{V}_6 = (1.634\vec{i} - 1.153\vec{j}) m/s$$

-Conservation of momentum.

$$\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t} \int_{CV} \vec{V} \rho dV + \int_{CS} \vec{V} \rho \vec{V} d\vec{S}$$

$\frac{\partial}{\partial t} \int_{CV} \vec{V} \rho dV$ is equal zero because is a steady flow.

Force in y-direction:

$$\vec{F}_y = \int_{CS} v \rho \vec{V} d\vec{S} = \int_{CS} v \rho \vec{V} d\vec{S} + \int_{CS} v \rho \vec{V} d\vec{S} = v_1 (-\rho V_1 A_1) + v_6 (\rho V_6 A_6) \rightarrow [v_1 = 0]$$

$$\vec{F}_y = (\rho V_6 A_6) = -1.153 \text{ m/s} \times (998.2 \frac{\text{kg}}{\text{m}^3} \times 2 \text{ m/s} \times 1.0937e-4 \text{ m}^2)$$

$$F_y = -0.25 \text{ N}$$

$$\text{Total torque} = -0.25 \text{ N} \times 19 \text{ mm} \times 4 \text{ flutes} = 19 \text{ N mm}$$

Results & Discussions.

This is likely the most important section of this project report. Here, we will find the summarized results of the experiments, simulations and calculations in the form of figures and tables. This results section includes three subsections:

- Analytical results.
- Theoretic results.
- Experimental results.

Simulation. Ansys (Fluent).

Two Dimensional simulation.

- Drag Coefficient.

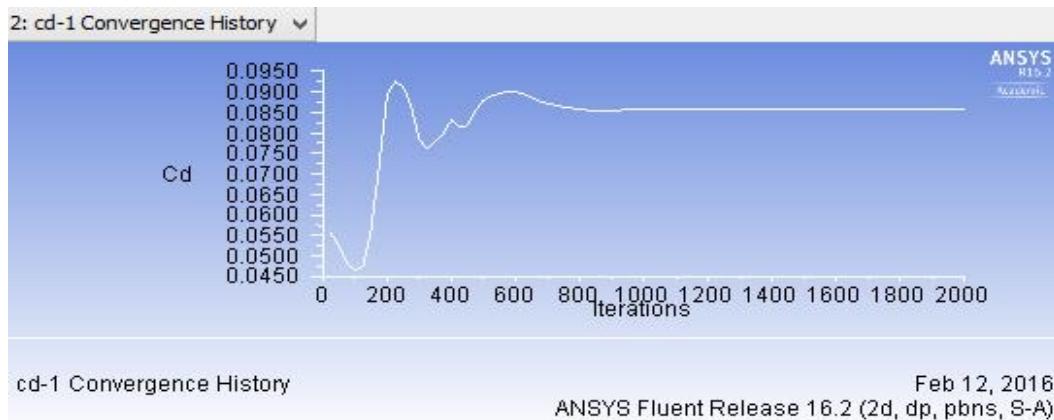


Illustration [61]

- Table of data.

iter	continuity	x-velocity	y-velocity	nut	C1-1	Cd-1
1950	5.7767e-06	2.5026e-08	1.0282e-08	1.0978e-07	2.1499e-03	8.5962e-02
1975	5.7590e-06	2.4561e-08	1.0062e-08	1.0086e-07	2.1491e-03	8.5962e-02
2000	5.7460e-06	2.4181e-08	9.8598e-09	9.4314e-08	2.1487e-03	8.5963e-02

Illustration [62]

- Velocity vectors visualization.

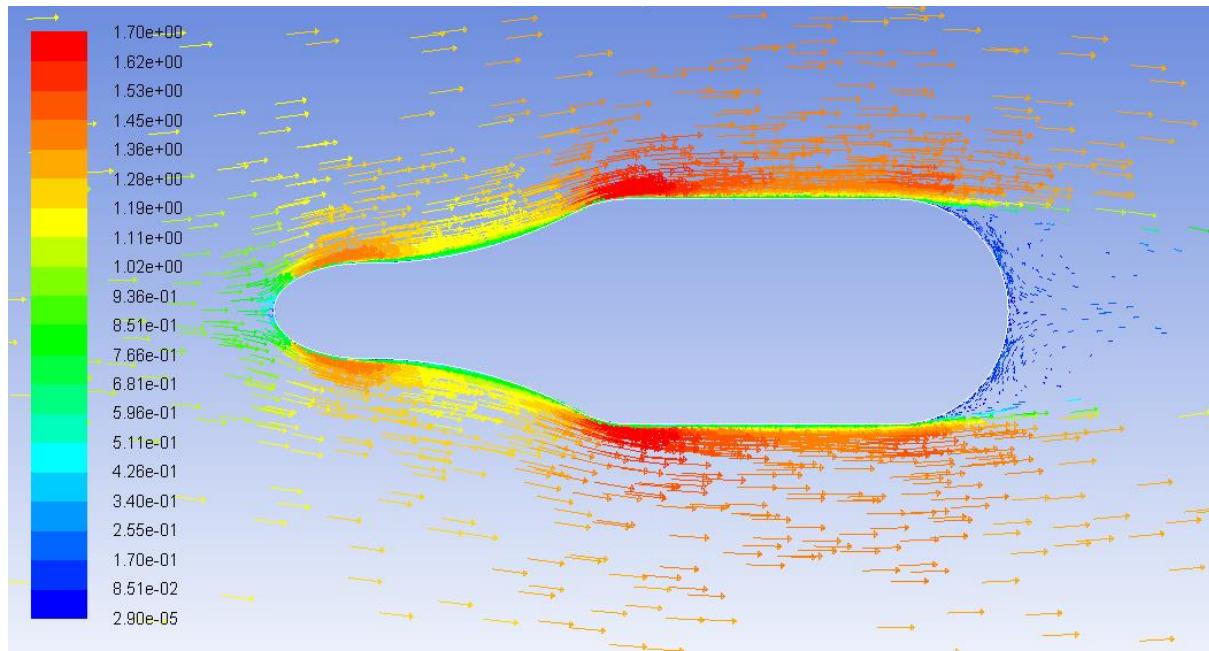


Illustration [63]

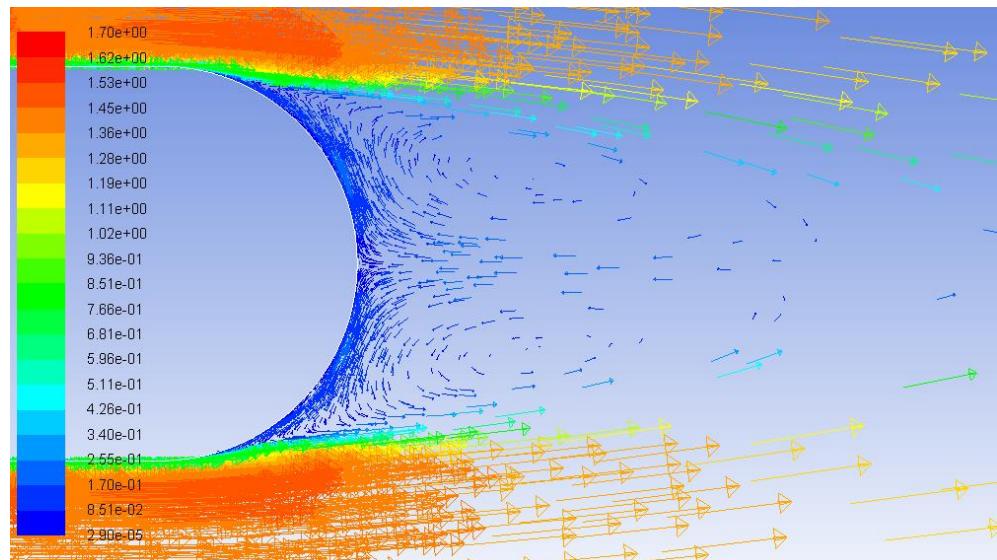


Illustration [64]

In these two illustrations, we can see the velocity vectors going through the shape. The shown behavior was easy to predict. We can see high velocity at concave parts of the shape and low velocity vectors at the convex parts of the shape. Also, right at the outlet it shows up vortexes producing low pressure at the outlet. These vortexes are an important factor of the drag forces.

- Comparison between Cylinder and MiniTap.

Finally, it is going to be plot the Cd coefficient of our device and cylinder on the same graph as a function of the Reynolds number. The result of this simulation could be predicted because the MiniPat geometry is a more streamline body than a cylinder. So, that why the drag coefficient is 0.1 time smaller than cylinder drag coefficient.

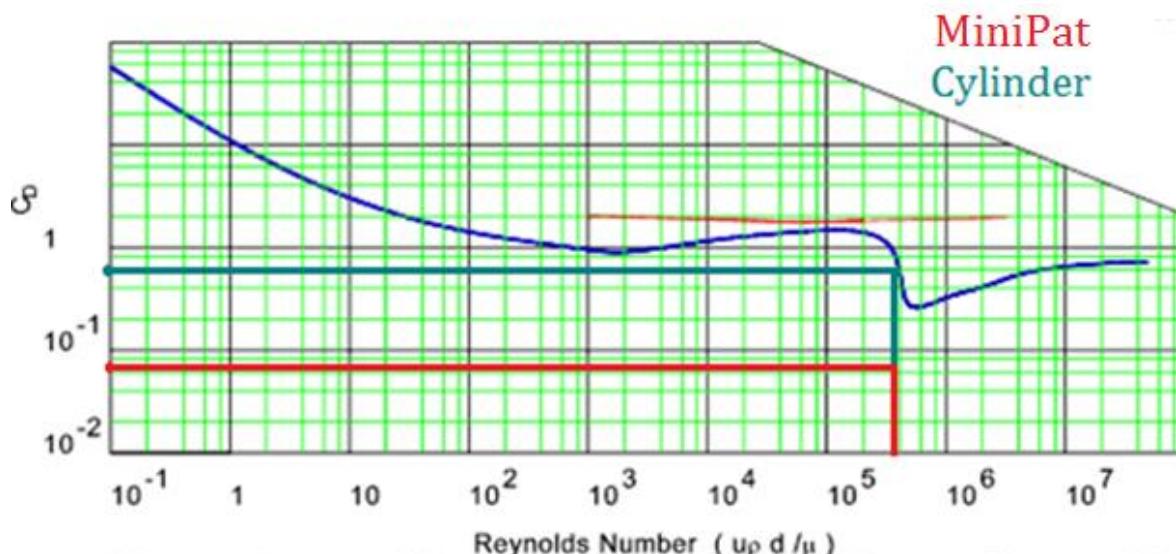


Illustration [65]

Three Dimensional simulation. Non-Fluted

Mesh N° One.

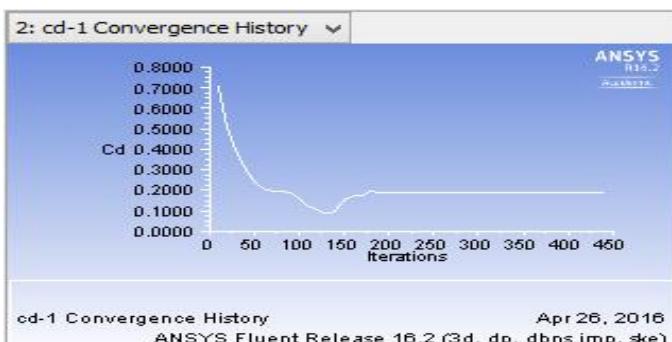
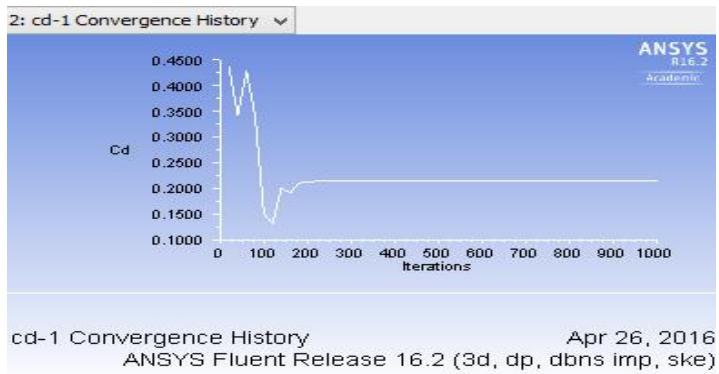


Illustration [66]

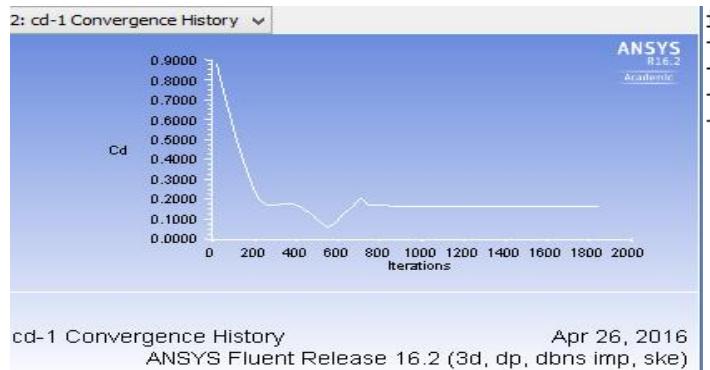
Mesh N° Two.



iter	C1-2	C1-1	Cd-1	time/iter
900	-1.9867e-02	-7.6972e-02	2.1391e-01	0:00:23 100
920	-1.9867e-02	-7.6972e-02	2.1391e-01	0:00:18 80
940	-1.9867e-02	-7.6972e-02	2.1391e-01	0:00:13 60
960	-1.9867e-02	-7.6972e-02	2.1391e-01	0:00:09 40
980	-1.9867e-02	-7.6972e-02	2.1391e-01	0:00:04 20
1000	-1.9867e-02	-7.6972e-02	2.1391e-01	0:00:00 0

Illustration [67]

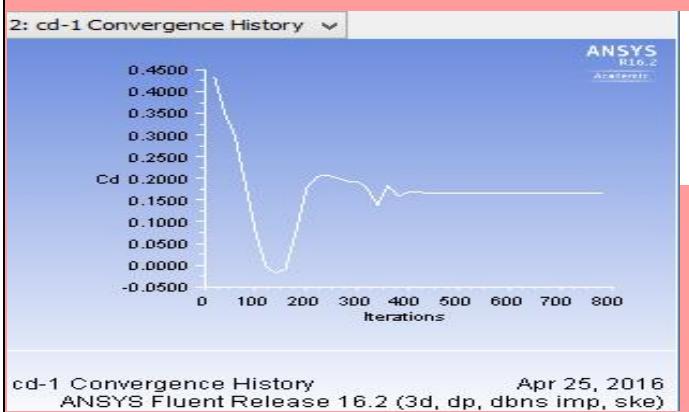
Mesh N° Three.



iter	C1-2	C1-1	Cd-1	time/iter
1780	-1.5592e-03	2.5493e-03	1.6573e-01	0:08:57 220
1800	-1.5595e-03	2.5501e-03	1.6573e-01	0:08:11 200
1820	-1.5599e-03	2.5508e-03	1.6573e-01	0:07:20 180
1840	-1.5603e-03	2.5513e-03	1.6573e-01	0:06:31 160

Illustration [68]

Mesh N° Four

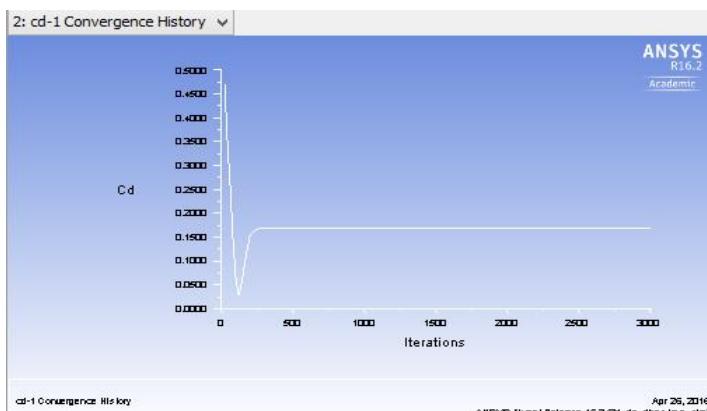


iter	C1-2	C1-1	Cd-1	time/iter
680	-1.3098e-03	-2.7000e-03	1.6668e-01	1:27:24 2000
700	-1.3151e-03	-2.6950e-03	1.6668e-01	1:25:43 1980
720	-1.3109e-03	-2.7011e-03	1.6668e-01	1:24:13 1960
740	-1.3089e-03	-2.6990e-03	1.6668e-01	1:22:51 1940
760	-1.3067e-03	-2.7007e-03	1.6668e-01	1:20:38 1920
780	-1.3106e-03	-2.6975e-03	1.6668e-01	1:19:40 1900

Selected mesh

Illustration [69]

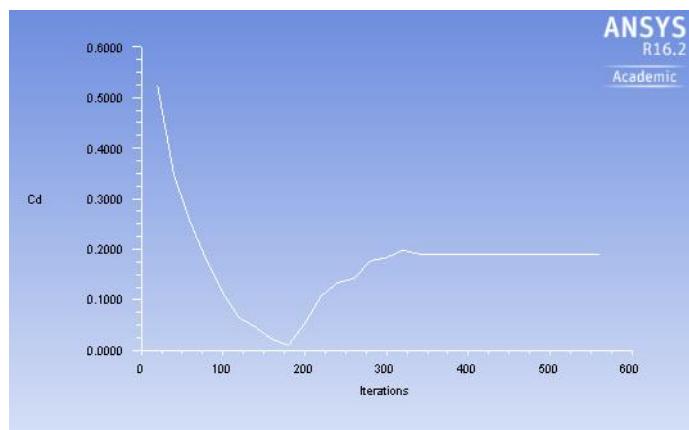
Mesh N° Five



iter	C1-2	C1-1	Cd-1	time/iter
2880	-4.6381e-03	-1.3321e-04	1.6941e-01	0:06:41 120
2900	-4.6379e-03	-1.3314e-04	1.6941e-01	0:05:34 100
2920	-4.6387e-03	-1.3394e-04	1.6941e-01	0:04:27 80
2940	-4.6385e-03	-1.3376e-04	1.6941e-01	0:03:21 60
2960	-4.6379e-03	-1.3285e-04	1.6941e-01	0:02:14 40
2980	-4.6382e-03	-1.3364e-04	1.6941e-01	0:01:07 20
3000	-4.6387e-03	-1.3392e-04	1.6941e-01	0:00:00 0

Illustration [70]

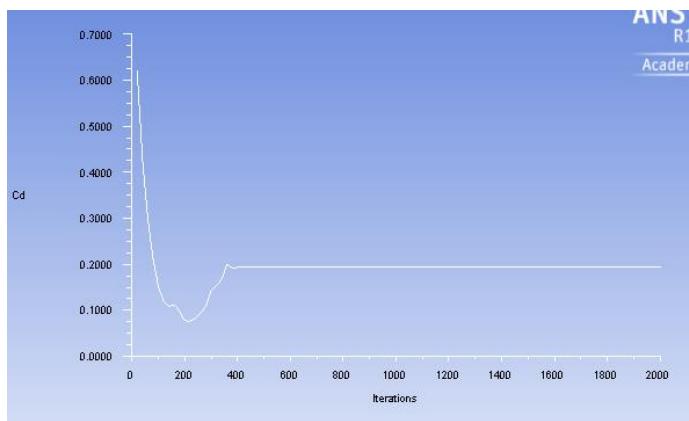
Sixth Mesh (corrected aspect ratio)



iter	C1-2	C1-1	Cd-1	time/iter
460	8.2205e-04	-2.1128e-04	1.9027e-01	2:33:19 1540
480	8.3356e-04	-2.1669e-04	1.9028e-01	2:31:43 1520
500	8.2663e-04	-2.1096e-04	1.9027e-01	2:29:31 1500
520	8.1470e-04	-1.9885e-04	1.9027e-01	2:28:37 1480
540	8.0424e-04	-1.9034e-04	1.9027e-01	2:28:11 1460
560	7.9808e-04	-1.8217e-04	1.9026e-01	2:25:58 1440

Illustration [71]

Seventh Mesh (corrected aspect ratio)



iter	C1-2	C1-1	Cd-1	time/iter
1780	8.3377e-05	2.7270e-04	1.9405e-01	0:58:34 220
1800	8.3409e-05	2.7273e-04	1.9405e-01	0:53:14 200
1820	8.3181e-05	2.7277e-04	1.9405e-01	0:47:54 180
1840	8.3453e-05	2.7290e-04	1.9405e-01	0:42:34 160

Illustration [72]

After these seven simulations, we have considered, which mesh can be an appropriate mesh for this project. The mesh number four gives us a good results and with a relatively low number of cells (726918). It is very important to let the reader know that, the more refined is the mesh the better results we can obtain, nevertheless this is not the only aspect to take into consideration. We also need to improve complex geometries zones in our mesh and set the mesh depending on the fluid behavior at different parts in the domain. To measure general quality of the mesh, we can use different methods, any case I strongly recommend, aspect ratio and orthogonal quality method.

Velocity Vectors at the wall.

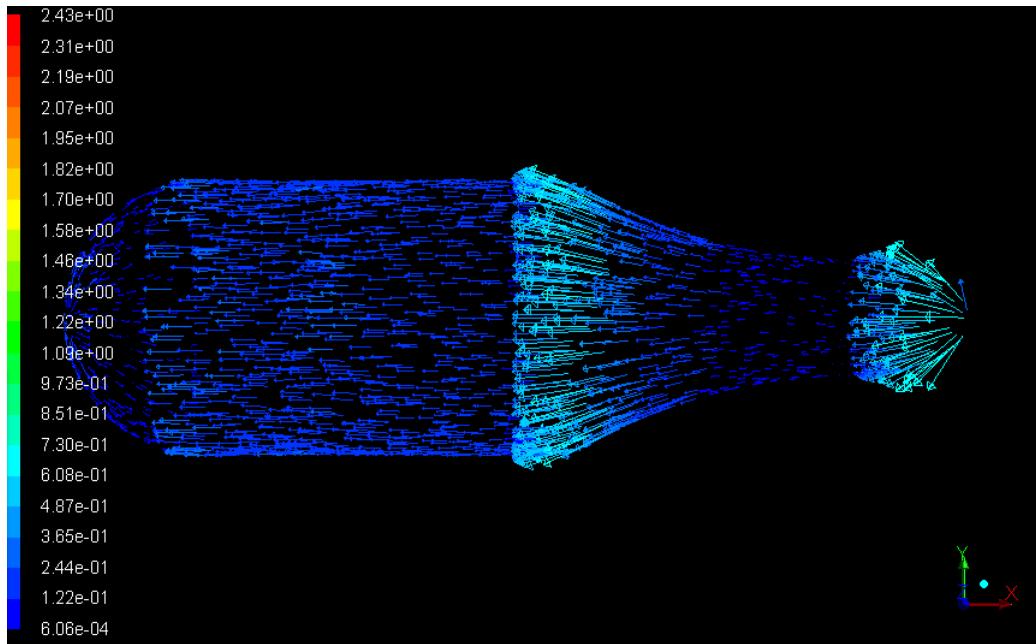


Illustration [73]

In the illustration above, it can be seen the velocities vector at the wall, same as we have commented before velocity vectors notably increase at the concave parts of the shape.

Below, we can appreciate the velocity vectors getting across the domain. The behavior of this flow is exactly the same as the 2D simulation, this is because the 3D simulation is revolution of the 2D simulation respect its main axis.

Velocity Vectors in the interior.

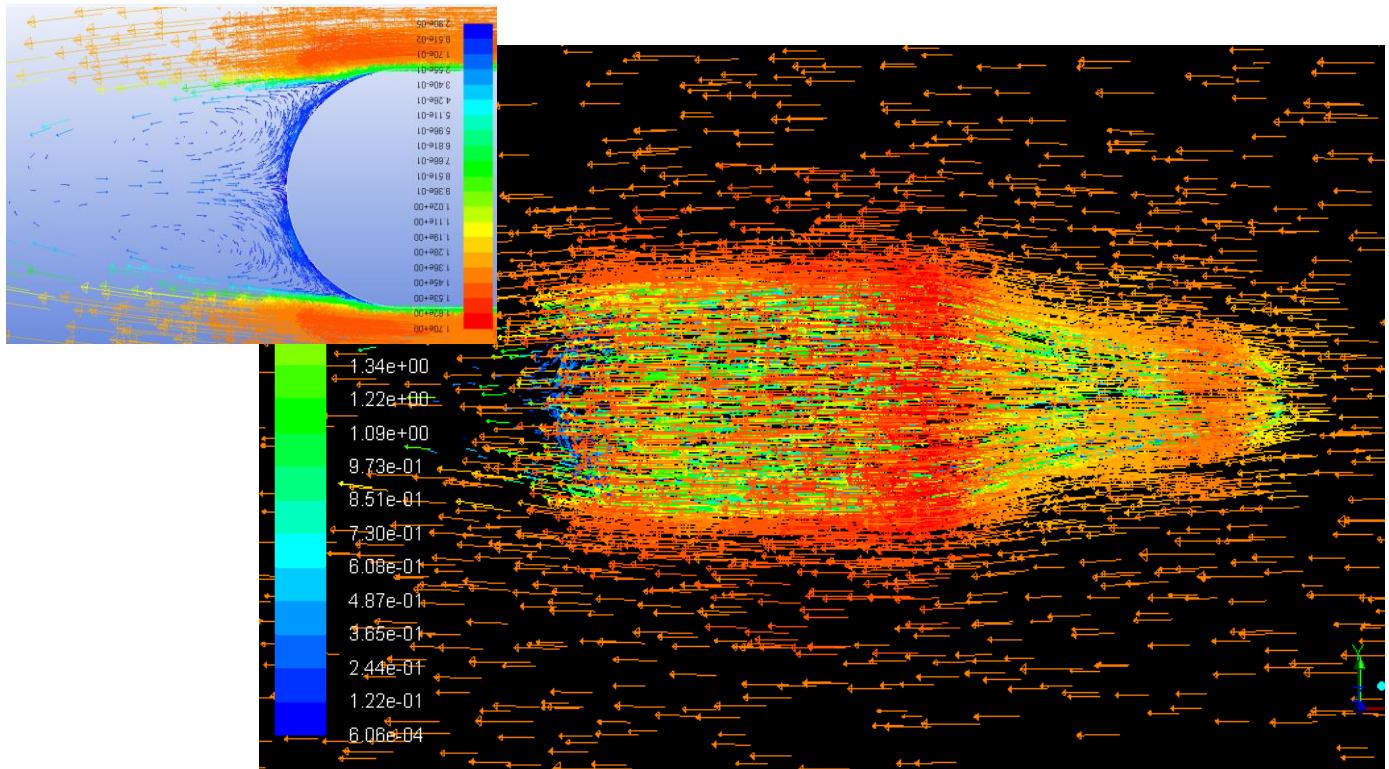


Illustration [74]

Pressure at the wall.

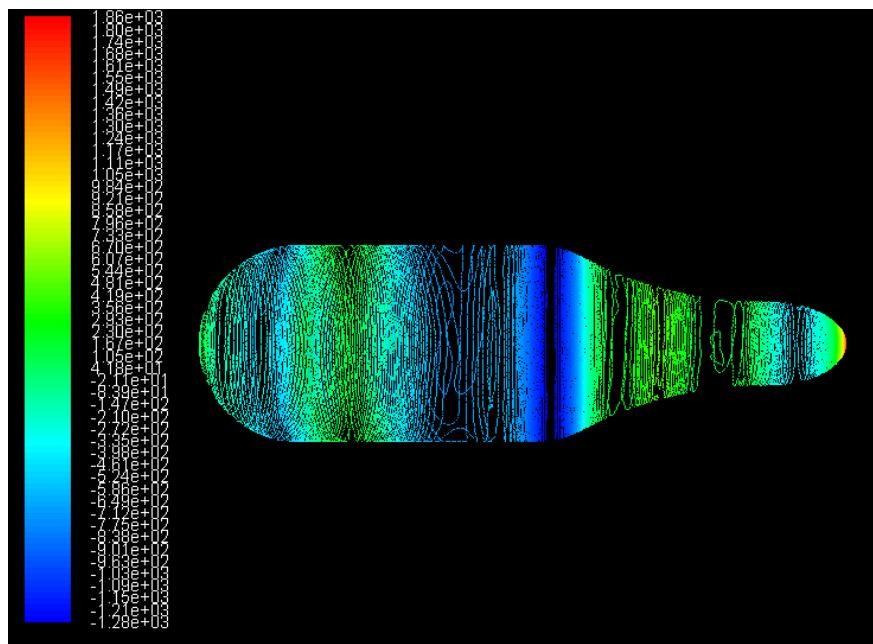


Illustration [75]

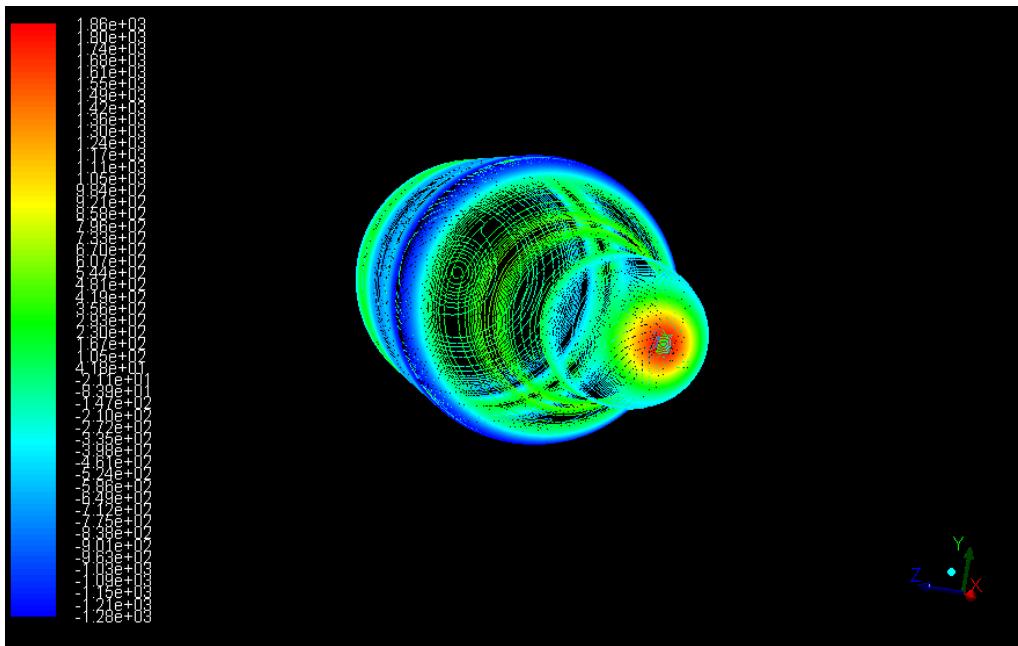


Illustration [76]

In the illustration [75] and [76], we can see the pressure contours at the wall. These images are very interesting because let us see, where the MiniPat is going to have the highest and the lowest pressure, this information can be useful at the time to design the flutes in the MiniPat. The highest pressure takes place at the front, (stagnation point). One more observation of these visualizations is how velocity vector and pressure contours fulfill the Bernoulli equation.

Three Dimensional simulation. Fluted.

As we have said in the method section of this project, the mesh, which it has been applied for the fluted shape simulation is quite similar to the fourth mesh from the non-fluted simulation.

Below, we see the results for this simulation:

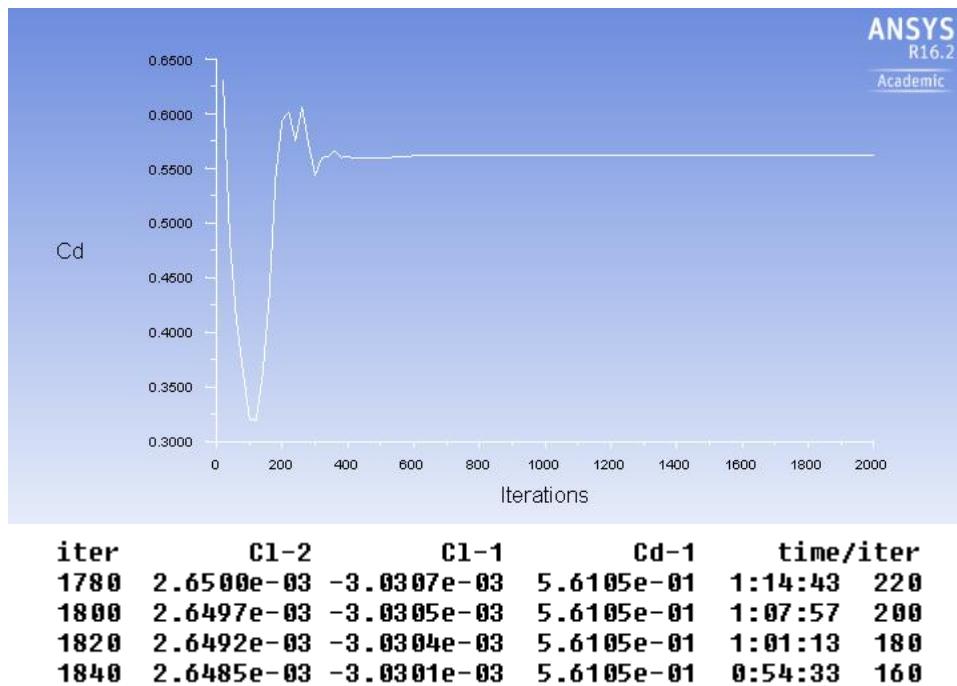


Illustration [77]

In further steps of this results section we will compare this result with the experimental results from the wind tunnel.

In the next illustration we can see the velocity vectors, as we briefly said in the methods sector, this model (model 2) was chosen instead of the model 1 because at the inlet of the flutes, the disruptions in the fluid are as minimal as possible.

With respect to pressure at the wall, the pressure contours seem very good over the shape. The highest pressure point is found it at the stagnation point and we can see pressure low the zero value at the outlet.

In the path-lines, we can see how the fluid acts on the MiniPat, when this remains fixed. (Static simulation).

Velocity Vectors at the wall.

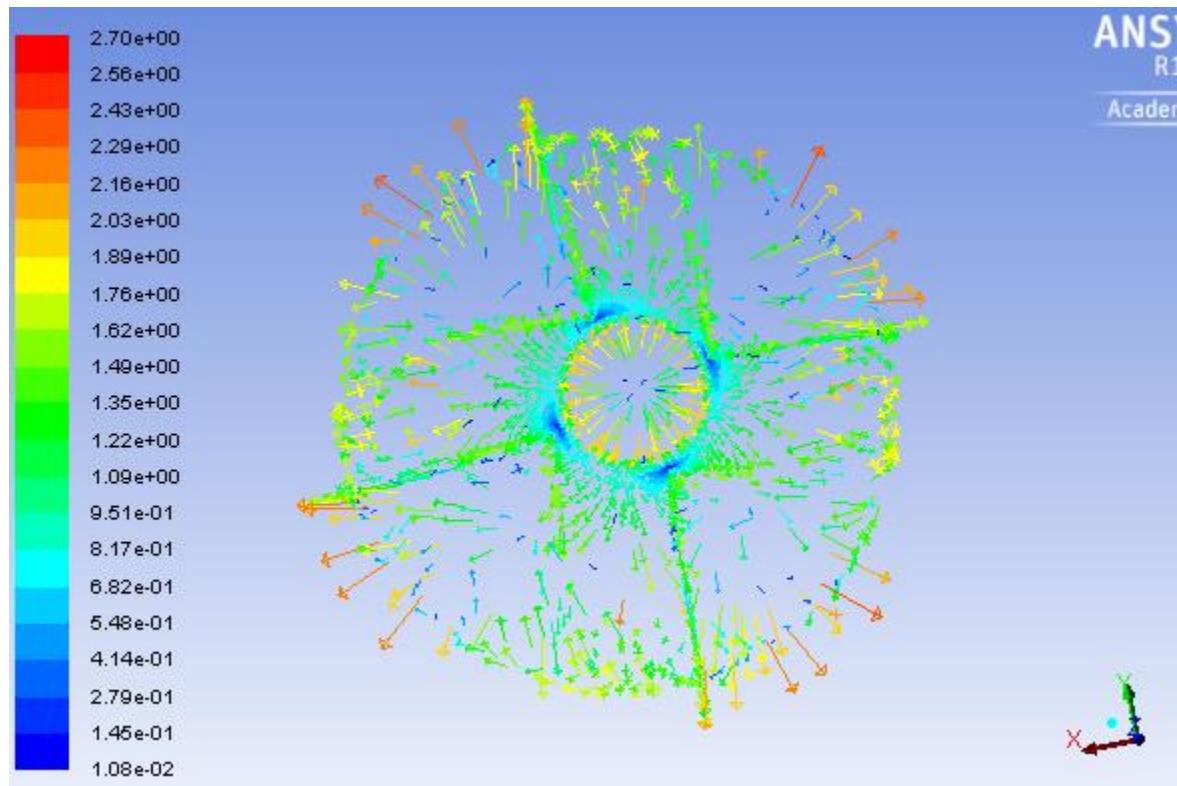


Illustration [78]

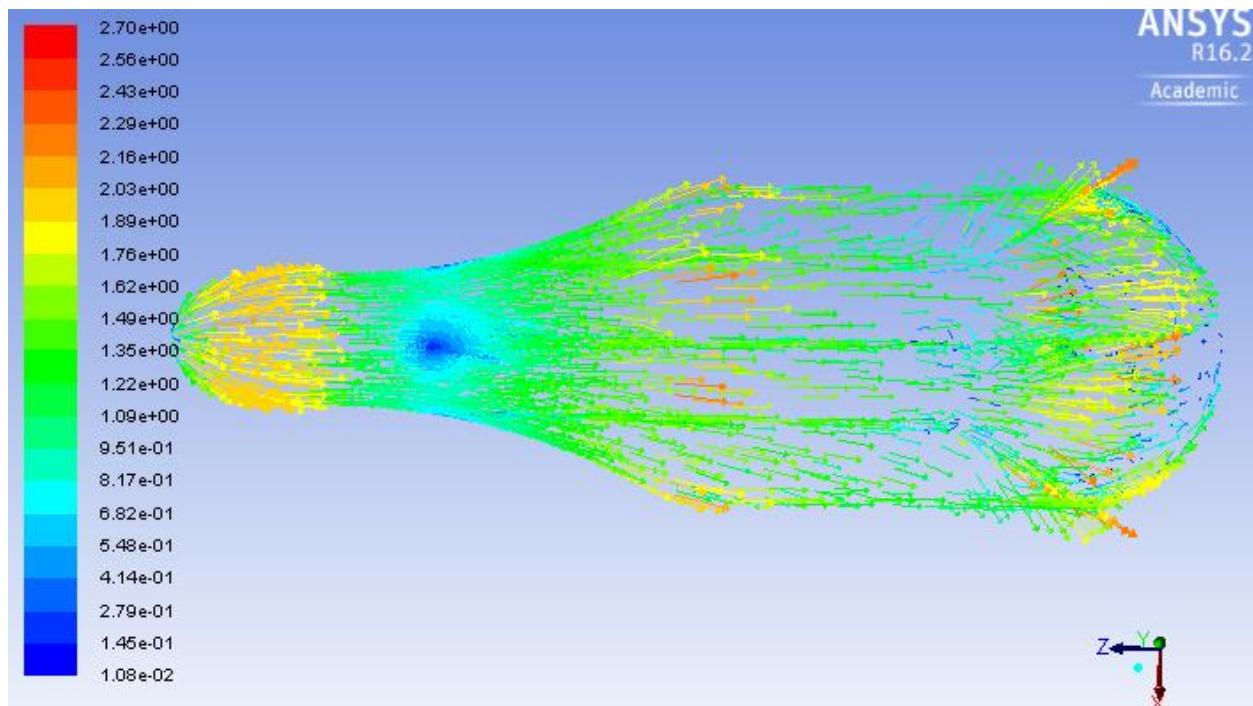


Illustration [79]

Pressure at the wall.

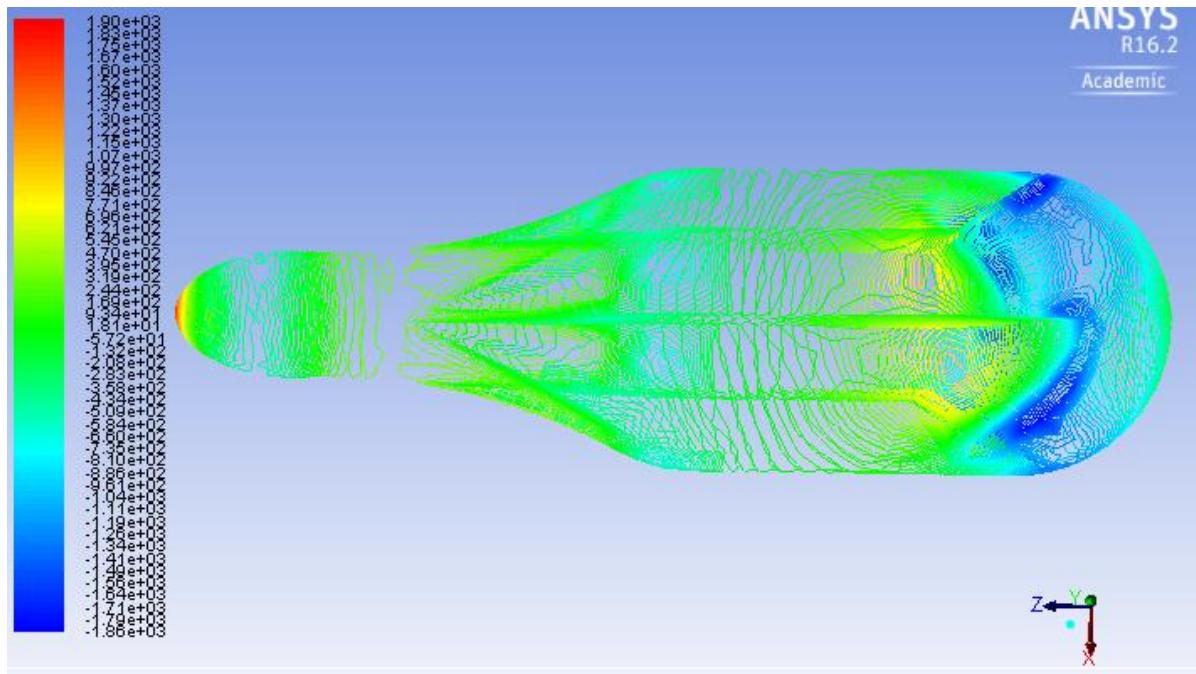


Illustration [80]

Paths-lines.

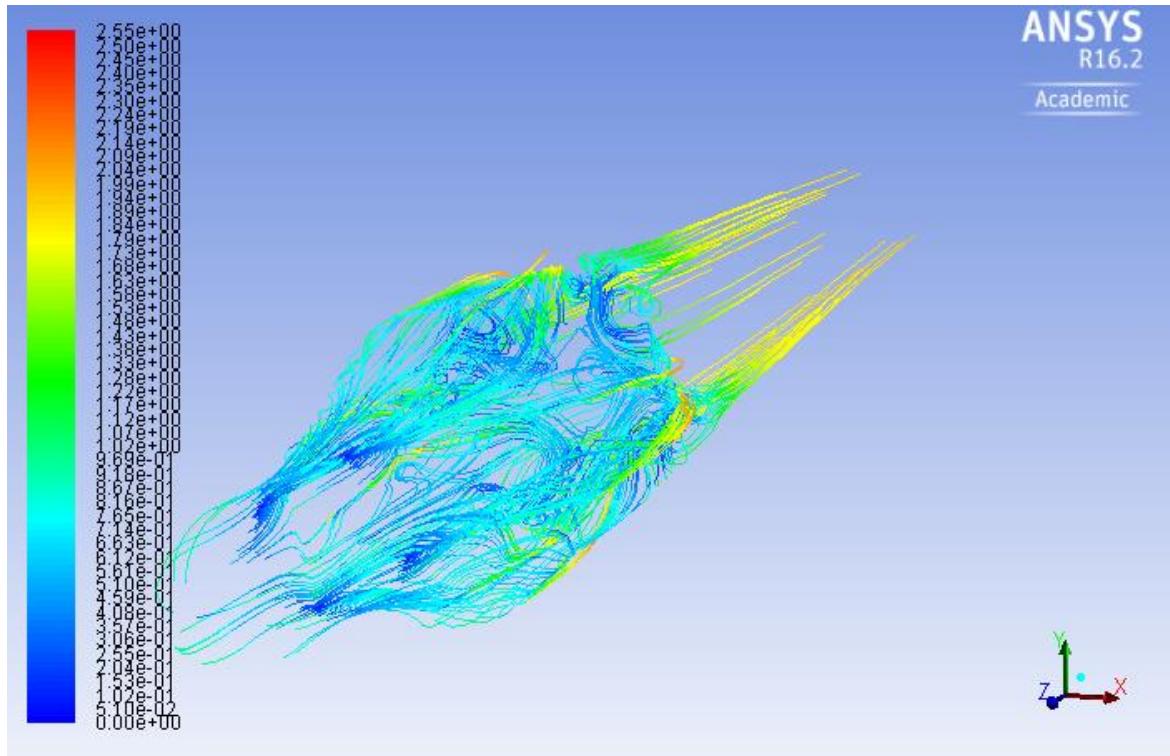


Illustration [81]

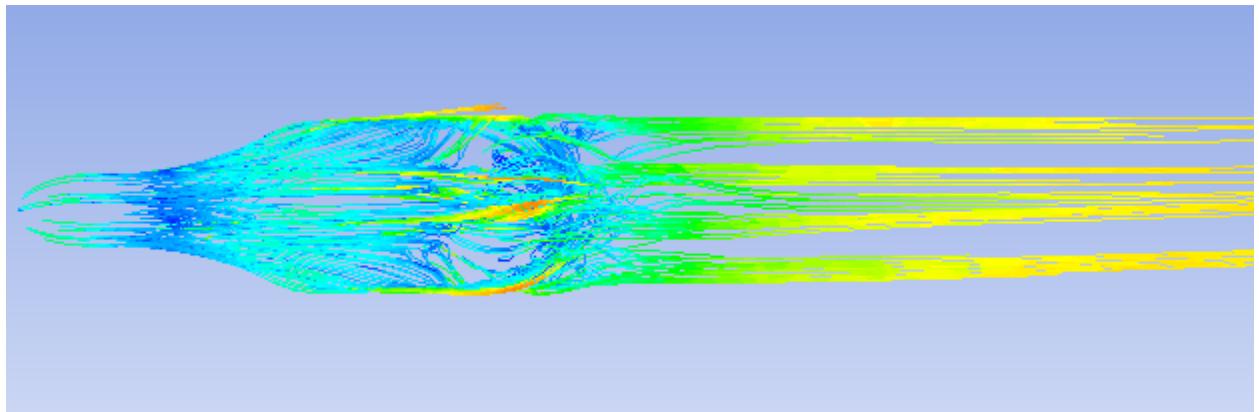


Illustration [82]

Visualization of the MiniPat rotating.

Velocity Vectors

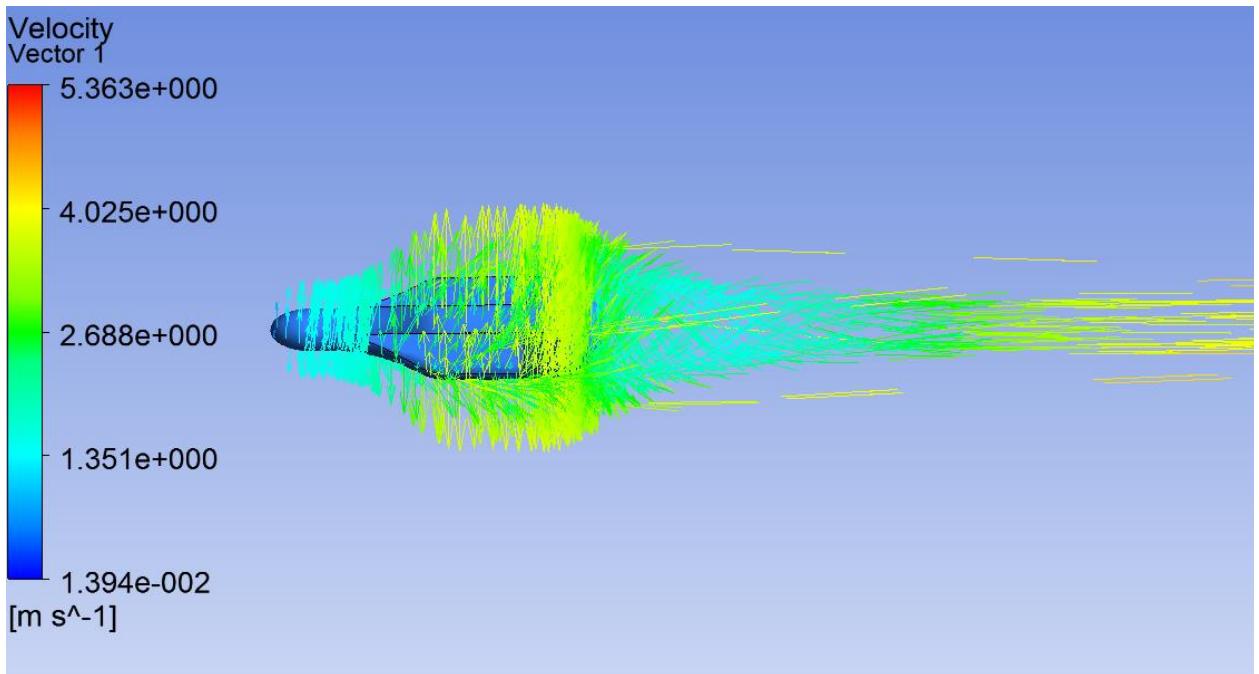


Illustration [83]

Velocity Streamline

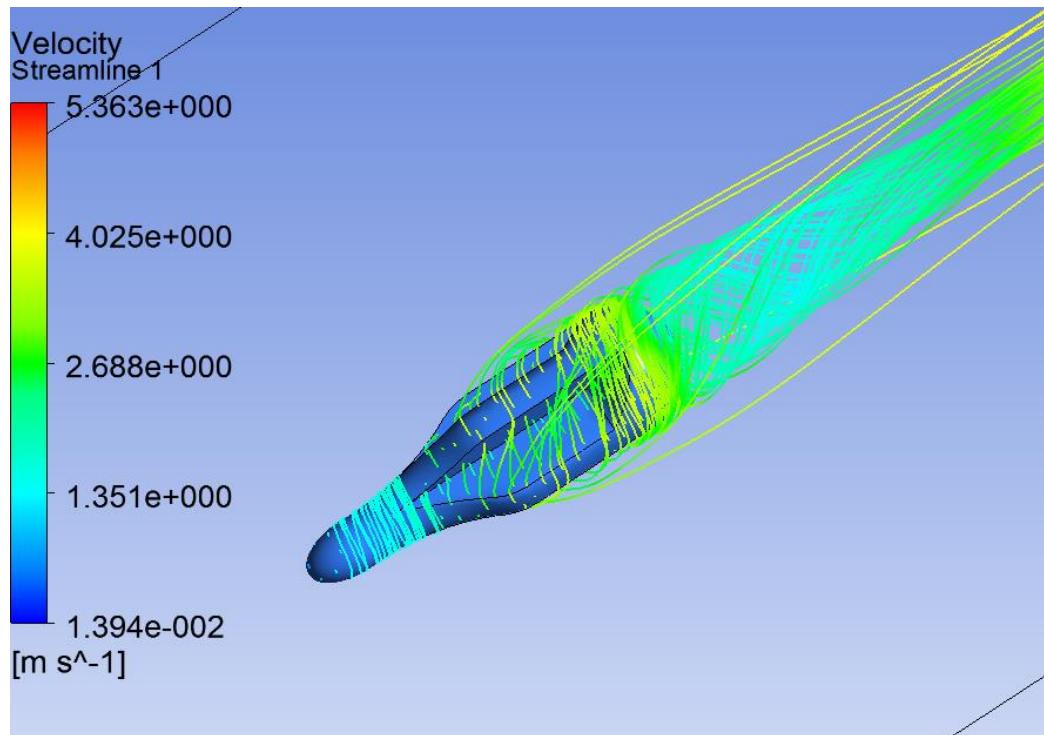


Illustration [84]

Paths-lines.

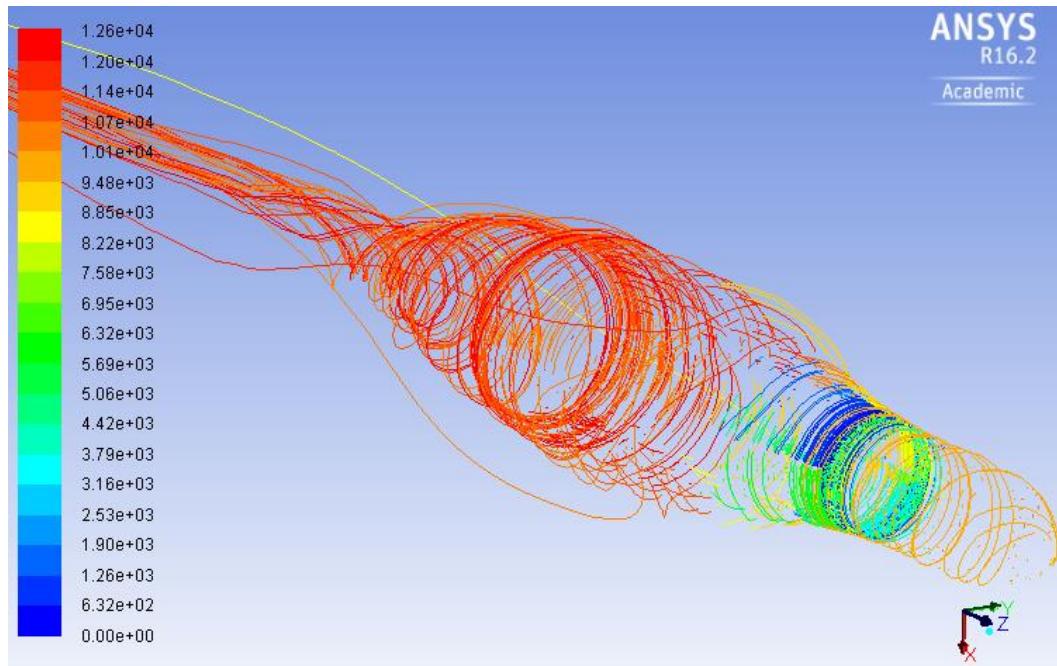
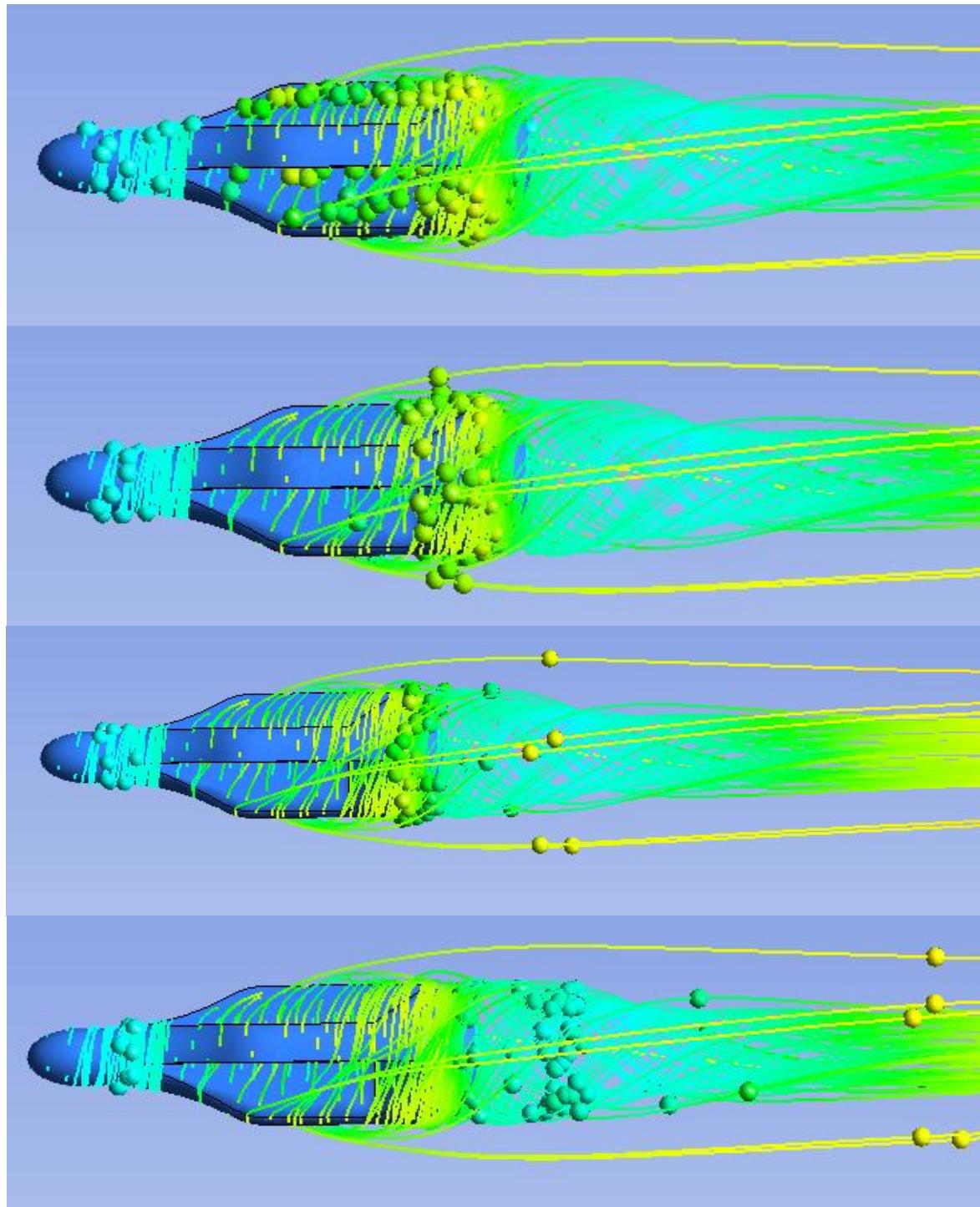


Illustration [85]

Particles evolution through the streamline.



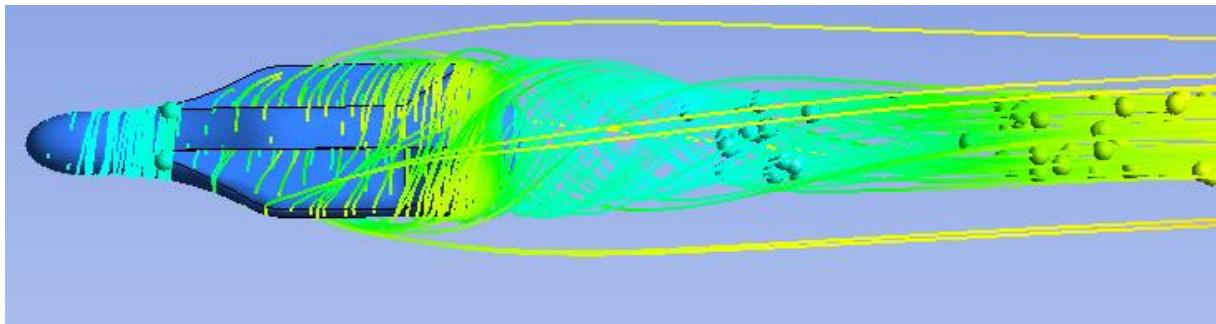


Illustration [86]

In this part we can see the rotational behavior of the MiniPat through a fluid. In the real case the fluid is going to make to rotate the MiniPat. For this simulation, we make the MiniPat to spin and this rotation makes the fluid go through the device. This is totally the same behavior but in an opposite way.

I would like to get across the last consecutive illustrations, where we can see very clear how the particles move in track with the streamlines.

As an observation and an important point of this model. We can see how the streamlines at the outlet are guided in a perpendicular direction respect to the MiniPat, this fact might produce unnecessary big vortexes at the outlet, as we can see. Moreover, this design only produce has just one force component which actually generate torque respect the main axis.

For future models, let's take into consideration the idea of designing the flutes with the aim to guide the fluid exactly behind the MiniPat and introduce a second force component which produce torque.

Torque.

In this table, which has been printed in Fluent, we can see the main component that produces torque in the MiniPat:

Forces - Direction Vector (1 0 0)			
	Forces (n)		
Zone	Pressure	Viscous	Total
fin1	0.076406631	-0.00060472632	0.075801905
Net	0.076406631	-0.00060472632	0.075801905

Illustration [87]

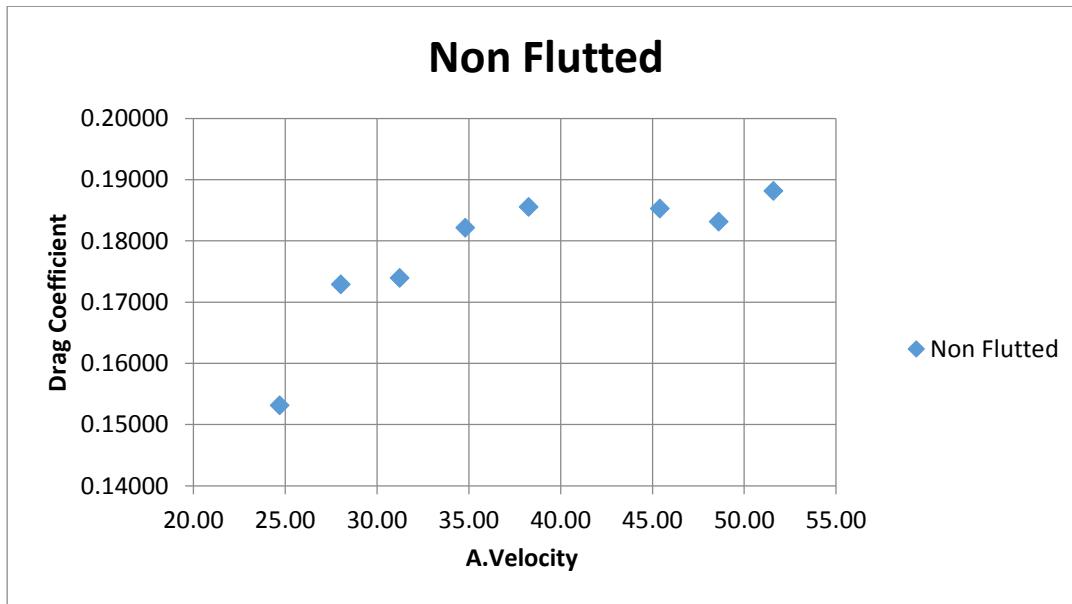
$$\text{Total Torque} = 19 \text{ mm} \times 0.0758 \text{ N} \times 4 \text{ flutes} = 5.8 \text{ N mm}$$

Experimental Results.

In this part of this section we are going to see the experimental results at the wind tunnel lab. These results are from the 1.5/1 scale model. We have set the corrected velocity in the software window and taken drag forces at different velocities. Finally we plot the drag coefficient as a function of velocity.

Non Flutted			
Wind Tunnel Velocity (m/s)	Actual Velocity (m/s)	Drag Force (N)	C_Drag
22.5	24.70	0.11	0.15316
25.3	28.03	0.16	0.17294
28	31.24	0.2	0.17398
31	34.81	0.26	0.18215
33.9	38.27	0.32	0.18556
39.9	45.41	0.45	0.18531
42.6	48.62	0.51	0.18317
45.1	51.60	0.59	0.18816

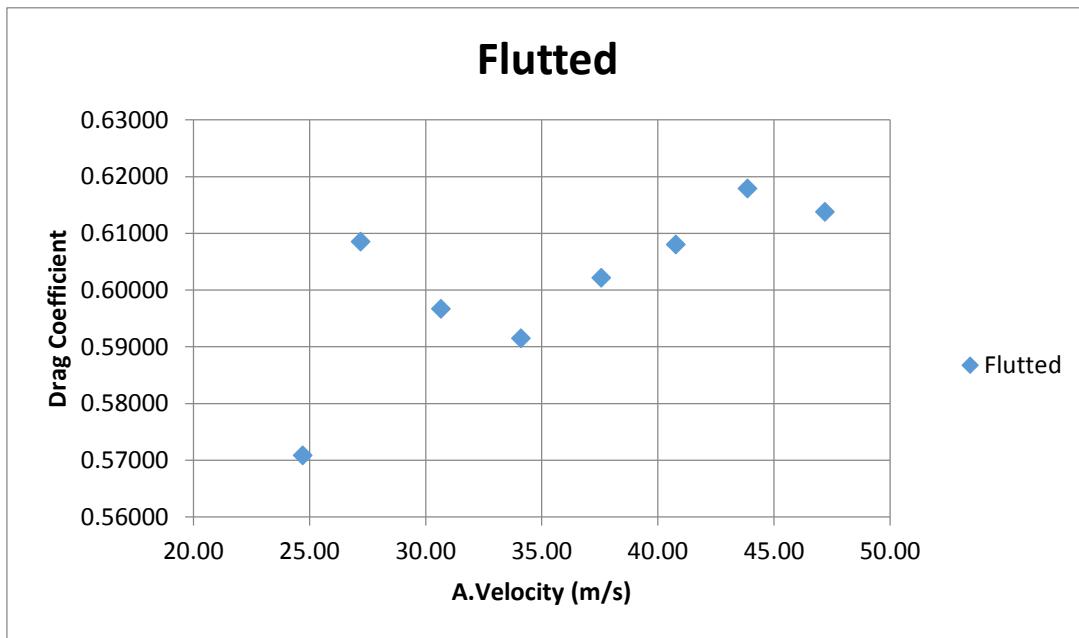
Table [8].



Plot [9]

Flutted			
Wind Tunnel Velocity (m/s)	Actual Velocity (m/s)	Drag Force (N)	C_Drag
22.5	24.70	0.41	0.57087
24.6	27.20	0.53	0.60851
27.5	30.65	0.66	0.59666
30.4	34.10	0.81	0.59150
33.3	37.55	1	0.60214
36	40.77	1.19	0.60801
38.6	43.86	1.4	0.61791
41.4	47.20	1.61	0.61377

Table [9].



Plot [10]

We can see from these result that for a different test from 24.7 m/s to about 50 m/s the drag coefficient remain quite similar, with a tendency to increase.

Now, if we compare experimental result with simulations, we can observe how close they are. Let's see the error between these results:

Non-Fluted MiniPat Error:

$$Error = \frac{0.15 - 0.166}{0.166} = 9.6 \%$$

Fluted MiniPat Error:

$$Error = \frac{0.57 - 0.561}{0.561} = 1.6 \%$$

These two error tell us how accurate are our simulations. Both of them are very good, working in the same order of magnitude and they are not extremely high. It is important to take into account that none of the experimental and numerical results are perfect, because the CFD result is an approximation to a theoretical result and experimental results are supposed to be result in a real case, however we need to be aware of AeroLab wind tunnel is only for educational aims.

Conclusions.

In this section we are going to conclude with a concise statement of the significant findings of the work.

So, we have seen the difficulties of simulating an unknown geometry, during this process we have learnt, it needs to have references values from experimental data of other researches, wind tunnel testes, approximations to other geometries... to compare, as long as it is possible. If we are not able to have reference values, we will be obligated to develop an independent study as we have done which might be complex.

After our independent study, we could match the drag coefficient with the experimental results from the wind tunnel. However, the rotatory MiniPat got a rod at the end, which is the bar of the forcemeter, with the aim to improve future simulation, we will have design this bar in our CFD model, then the results of our simulation are going to be more accurate respect the experimental data.

To calculate the angular speed and torque, we might test it in different ways:

- We can install the rotatory MiniPat at the wind tunnel with a couple of bearings and with an electric motor which is going to be rotated by the MiniPat. So, the angular speed can be calculated by strobe light and we can obtain the generated power produce by the MiniPat. The torque is going to be a result of dividing the power by the angular speed. On the other hand the torque can be easily calculate with a torque sensor.
- We can also test the MiniPat in a transparent water tank. The idea for this experiment is leave the MiniPat from the top of the tank to the bottom. The rotatory MiniPat is going to descend and spin through the tank by gravitational forces. Using the water parameter, how long it takes the MiniPat to get the bottom and how many revolutions the MiniPat does, we can obtain the experimental results that we are looking for.

I would like to comment how we can improve the design of the model. In the visualization of the streamlines we can observe how the streamlines abruptly interact with the freestream fluid at the outlet of the MiniPat. To make this interaction smoother we can design the outlet of the MiniPat with the aim to exhaust the fluid right after the outlet instead of the lateral of the MiniPat.

Looking into the future, I strongly recommend to apply these improvement. The objective will be to install a generator and GPS in the rotatory turbine in this way we can achieve a MiniPat with an unlimited living time.

References:

1. J. Pritchard. "Fox and McDonald's introduction to Fluid Mechanics", 8th edition, 2011.
2. John Anderson Jr. "Fundamentals of Aerodynamics", 5th edition, 2010.
3. John Anderson Jr. "Computational Fluid Dynamics. The basic with applications", 1st edition, 1995.
4. Richard S. Figliola\ Donald E. Beasley." Theory and Design for Mechanical Measurements", 5th edition, 2011.
5. Ansys, Customer Training Material."Introduction to ANSYS Workshop", 2010.
6. Ansys, Customer Training Material."Introduction to ANSYS FLUENT", 2010.
7. AeroLab Handouts, "Manuel of AeroLab"
8. Wildlife Computers," ProductSheet_MiniPAT_v15-10", 2014.
9. "Test by Yachting Monthly: "Highest Top Speed..."" *Folding Propellers*. N.p., n.d. Web. 10 May 2016.
10. "RoyMech Index Page." RoyMech Index Page. N.p., n.d. Web. 10 May 2016
11. NASA. NASA, n.d. Web. 10 May 2016.
12. "CFD Online." CFD Online. N.p., n.d. Web. 10 May 2016
13. "Local Weather Conditions - Northern Arizona University, Flagstaff, Arizona."Local Weather Conditions - Northern Arizona University, Flagstaff, Arizona. N.p., n.d. Web. 10 May 2016.

Appendix

Here we can find the plans for the two rotatory MiniPat models.