Politecnico di Torino
Corso di Laurea Magistrale
in Ingegneria Aerospaziale

Tesi di Laurea Magistrale

CFD Streamlined Bodies Simulation and Optimization for Natural Laminar Flow provided by STAR-CCM+ and HEEDS softwares

Relatore: Prof. Paolo Baldissera

Candidato: Durante Leonardo

Marzo 2019
Eccoci arrivati alla parte più letta della tesi. Dimeneticherò sicuramente qualcuno ma vi assicuro che il lavoro dietro le quinte di questo lungo, lunghissimo, immenso viaggio (ho reso l’idea della lunghezza?!?) è stato grandioso ed i miei ringraziamenti vanno a tutti voi.

Innanzitutto dedico un grazie a mamma e papà che continuamente mi hanno spronato ad andare avanti a suon di: ”Se ce l’ha fatta tuo padre”; oppure con: ”Sai chi si è laureato?”, ed altre frasi ”motivanti”. Scherzi a parte grazie per il sostegno di questi anni e...lo sapete è inutile che lo scriva. Visto che bel figlio che avete cresciuto? :)

Un grazie inoltre va ai due moschettieri Emanuele e Ludovica (ludo tu leggi Ludovica ed Emanuele) che...sinceramente non c’è un motivo principale per ringraziarli talmente tante sono le cose per cui dovrei farlo! P.S. siete i miei fratelli preferiti!

Un grazie anche a quei tre disgraziati di Nunzio, Andrea e Davide (rigorosamente in ordine alfabetico) compagni di mille champions e battaglie.

Acknowledgments

Eccoci arrivati alla parte più letta della tesi. Dimeneticherò sicuramente qualcuno ma vi assicuro che il lavoro dietro le quinte di questo lungo, lunghissimo, immenso viaggio (ho reso l’idea della lunghezza?!?) è stato grandioso ed i miei ringraziamenti vanno a tutti voi.

Innanzitutto dedico un grazie a mamma e papà che continuamente mi hanno spronato ad andare avanti a suon di: ”Se ce l’ha fatta tuo padre”; oppure con: ”Sai chi si è laureato?”, ed altre frasi ”motivanti”. Scherzi a parte grazie per il sostegno di questi anni e...lo sapete è inutile che lo scriva. Visto che bel figlio che avete cresciuto? :)

Un grazie inoltre va ai due moschettieri Emanuele e Ludovica (ludo tu leggi Ludovica ed Emanuele) che...sinceramente non c’è un motivo principale per ringraziarli talmente tante sono le cose per cui dovrei farlo! P.S. siete i miei fratelli preferiti!

Un grazie anche a quei tre disgraziati di Nunzio, Andrea e Davide (rigorosamente in ordine alfabetico) compagni di mille champions e battaglie.
Abstract

The main topic of this thesis is the solution at engineering problem of minimum drag axisymmetric vehicles design. Drag reduction is only possible through manipulation of the vehicle shape, in order to delay viscous layer separation point. An affusolage body will be invested at zeros angle by a costant speed flow. In the first chapters is proposed a study of the physics problem, with an analysis of minimum drag bodies at difference Reynolds numbers, the boundary layer’s evolution across body’s length and it separation. Later is proposed CFD analysis with Gamma-Re\theta transition model using STAR-CCM+ and a subsequent optimization of the body’s shape thanks to HEEDS software.
Contents

Acknowledgments .................................................. ii

List of Figures .................................................. III

List of Tables .................................................. vi
  Introduction ................................................... vii
  Team Policumbent ............................................. vii

1 Sate Of Art .................................................. 1
  1.1 World’s Fastest Human-Powered Bike: Eta .................. 1
  1.2 Drag Coefficient and Reynolds Number .................... 5

2 Physics problem ............................................. 8
  2.1 Aerodynamic Drag .......................................... 8
  2.2 Laminar and Turbulent Flow ................................ 10
  2.3 Boundary Layers and Separation .......................... 12
    2.3.1 Transition Uncertainty ............................... 14

3 STAR-CCM+ .................................................. 15
  3.1 Finite Element Method .................................... 15
  3.2 Geometry .................................................. 17
    3.2.1 CAD-Models: B-2019 ................................. 17
    3.2.2 Gallery ............................................... 19
  3.3 Physics Condition ......................................... 20
    3.3.1 All y+ Wall Treatment ............................... 20
    3.3.2 Gamma Re-Theta Transition Model .................. 22
    3.3.3 Free Stream Edge ..................................... 23
List of Figures

1. Pulse (2011/2012) ........................................ vii
2. S-Trike (2013/2014) ....................................... viii
3. PulsaR (2014/2016) ....................................... viii
4. Taurus (2017-Today) ........................................ viii

1.1 Eta .......................................................... 3
1.2 Eta Inside .................................................... 3
1.3 Eta top View ............................................... 4
1.4 Drag Coefficient versus Reynolds Number ............ 5
1.5 Shaping Optimization versus Reynolds Number ....... 6
1.6 Optimized Body Design Regime I ......................... 7
1.7 Optimized Body Design Regime II ....................... 7
1.8 Optimized Body Design Regime III ...................... 7
1.9 Optimized Body Design Regime IV ....................... 7

2.1 Pressure and shear forces acting on body surface .... 9
2.2 Laminar vs Turbulent Velocity Profile ................ 10
2.3 Velocity Profile ........................................... 12
2.4 Convergence-Divergence Nozzle ....................... 13
2.5 Separation Point .......................................... 13

3.1 Geometry Body - Bad Mesh - Good Mesh ............. 16
3.2 B-2019 Front and Top Views ........................... 18
3.3 Wind Gallery .............................................. 19
3.4 Non-dimensional velocity as a function of across the three sublayers 20
3.5 low-y+ approach ........................................... 21
3.6 13-in Profile ............................................... 25
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.7 Field Function</td>
<td>26</td>
</tr>
<tr>
<td>3.8 Star CCM+ Drag Coefficient vs Experimental CD</td>
<td>27</td>
</tr>
<tr>
<td>3.9 Liquid crystal flow vs Star CCM+ visualization for free transition. Re=1.2E6</td>
<td>27</td>
</tr>
<tr>
<td>3.10 Boundaries</td>
<td>32</td>
</tr>
<tr>
<td>3.11 Volume Mesh</td>
<td>33</td>
</tr>
<tr>
<td>3.12 16 Prism Layers Details</td>
<td>33</td>
</tr>
<tr>
<td>3.13 Trail Details</td>
<td>34</td>
</tr>
<tr>
<td>3.14 Residuals</td>
<td>35</td>
</tr>
<tr>
<td>3.15 Drag</td>
<td>35</td>
</tr>
<tr>
<td>3.16 Wall Shear Stress</td>
<td>36</td>
</tr>
<tr>
<td>3.17 Wall-(y^+)</td>
<td>36</td>
</tr>
<tr>
<td>3.18 Speed and Pressure</td>
<td>37</td>
</tr>
<tr>
<td>3.19 Total Pressure</td>
<td>38</td>
</tr>
<tr>
<td>4.1 Design A - Body</td>
<td>40</td>
</tr>
<tr>
<td>4.2 Design A Residuals</td>
<td>41</td>
</tr>
<tr>
<td>4.3 Design A Drag</td>
<td>41</td>
</tr>
<tr>
<td>4.4 Design A Wall Shear Stress</td>
<td>41</td>
</tr>
<tr>
<td>4.5 Design B - Body</td>
<td>42</td>
</tr>
<tr>
<td>4.6 Design B Residuals</td>
<td>43</td>
</tr>
<tr>
<td>4.7 Design B Drag</td>
<td>43</td>
</tr>
<tr>
<td>4.8 Design B Wall Shear Stress</td>
<td>43</td>
</tr>
<tr>
<td>4.9 Design B Speed and Pressure</td>
<td>44</td>
</tr>
<tr>
<td>4.10 Design B Total Pressure</td>
<td>44</td>
</tr>
<tr>
<td>4.11 Design C - Body</td>
<td>45</td>
</tr>
<tr>
<td>4.12 Design C Drag and Residuals</td>
<td>46</td>
</tr>
<tr>
<td>4.13 Design C WSS and Wally+</td>
<td>46</td>
</tr>
<tr>
<td>4.14 Design C Speed and Pressure</td>
<td>47</td>
</tr>
<tr>
<td>4.15 Design C Total Pressure</td>
<td>47</td>
</tr>
<tr>
<td>5.1 Ground Prism Layer Volume Mesh</td>
<td>50</td>
</tr>
<tr>
<td>5.2 GE Drag</td>
<td>51</td>
</tr>
<tr>
<td>5.3 GE Wall Shear Stress</td>
<td>51</td>
</tr>
<tr>
<td>5.4 GE Wall-(y^+)</td>
<td>52</td>
</tr>
</tbody>
</table>
5.5 Four Splines ............................................................... 53
5.6 No AXB WSS .............................................................. 54
5.7 Maximum Diameters ..................................................... 54
# List of Tables

1. WHPSC results ............................................ vii

2. B-2019 Parameters .......................................... 17

3. Gallery Dimensions .......................................... 19

4. Gamma ReTheta Parameters .................................. 22

5. 13-Inches Profile Plot Data .................................. 24

6. Air Data .................................................. 29

7. Boundaries ................................................. 32

8. Mesh Conditions ............................................. 34

9. HEEDS Parameters ............................................ 39

10. Design A - Parameters ........................................ 40

11. Design B - Parameters ........................................ 42

12. Design C - Parameters ........................................ 45

13. Design Comparison ........................................... 48

14. Region Boundaries Condition ................................. 49
Introduction

Team Policumbent

This thesis born according to Policumbent team’s needs to reduce drag force for its human powered vehicles prototypes. Policumbent is a students Team from Politecnico of Torino, orientated to the design and the realization of human powered vehicles and to ride the created prototypes [2]. Since 2009 Policumbent team continually develop new models, from C.O.R.A. (Cycling Optimized Recumbent Aeroshape) (2009-2010), to the most recently Taurus (2017-today). Since 2015 they are trying to overcome human speed limits in the middle of Nevada desert at about 1400 meters above sea level in the annual World Human Powered Speed Challenge (WHPSC), reaching very good places against other Universitary Team and also to overcome italian speed record twice in 2016 (126.90 km/h) and 2018 (133.26 km/h). In the Run mode Cyclists have to drive 5 miles in order to reach their top speed and the 200 m long timing zone.

<table>
<thead>
<tr>
<th>Year</th>
<th>Prototype</th>
<th>Speed (km/h)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2015</td>
<td>Pulsar</td>
<td>116.19</td>
</tr>
<tr>
<td>2016</td>
<td>Pulsar</td>
<td>126.90</td>
</tr>
<tr>
<td>2017</td>
<td>Taurus</td>
<td>122.32</td>
</tr>
<tr>
<td>2018</td>
<td>Taurus</td>
<td>133.26</td>
</tr>
</tbody>
</table>

Table 1: WHPSC results

Figure 1: Pulse (2011/2012)
Figure 2: S-Trike (2013/2014)

Figure 3: PulsaR (2014/2016)

Figure 4: Taurus (2017-Today)
Chapter 1

Sate Of Art

Drag reduction by laminarization of the boundary layer plays an important role in aerodynamic aircraft design. For example, the current sailplanes high performances can only be obtained by extensive laminar flow regions on suction and pressure sides of the wing. With the application of laminar airfoil sections the drag contribution of the fuselage will achieve a significant amount.\[1\] Laminar to turbulent transition is a complex and yet not fully understood phenomenon.

1.1 World’s Fastest Human-Powered Bike: Eta

Eta is the World’s fastest human-powered bike with a maximum speed of 144 Km/h. The bike, named Eta for the Greek symbol used to denote efficiency in engineering, uses a highly aerodynamic shape and coating, an ergonomic reclining position for the rider, and modern composite materials such as carbon fiber weaves to provide as much power transfer as possible through the stiff bike frame.\[3\] The most important part of making the vehicle faster than an ordinary bicycle is by eliminating flow separation over the body. Doing this can decrease the drag over a given size object by over an order of magnitude!

To design the pressure profiles of the vehicle to eliminate flow separation, the Stratford-Smith criterion was used. This criterion is an analytic formulation to determine when the flow is on the verge of separation based upon the pressure gradient along streamlines.
1 – State Of Art

Once the separation pressure drag over the vehicle is eliminated, the shear surface
drag becomes the largest drag component we see on the vehicle. This is the drag
acting tangentially to the surface of the vehicle caused by the viscosity of the air
moving across it. The shear drag on the vehicle is determined by the state of the
boundary layer over the surface. The boundary layer grows to about a centimetre
thick at the trailing edge of our vehicles and can accounts for the vast majority of
our aerodynamic drag. The boundary-layer begins laminar at the nose but may
eventually destabilize into a turbulent boundary layer which has many times more
drag.

To design for laminar flow, a favourable pressure gradient (decreasing pressure)
along surface streamlines is desirable to assist in the stability of the laminar bound-
dary layer and prevent it from transitioning to turbulence. In the laminar region,
the shape of the pressure profiles are modelled based upon two-dimensional air-
foils are designed for extensive runs of laminar flow (most notably NACA 6-Series
airfoils). These typically follow a constant favourable (negative) pressure gradient
which allows for a similar amount laminar flow across various angles of attack.

Another thing to mention about extensive runs of laminar flow is its contribution
to reducing flow separation. Greater amounts of laminar flow substantially reduces
the momentum boundary-layer thickness (a measure of how much of the flow’s mo-
mementum is trapped in the boundary-layer) and allows for a shorter tail and a more
aggressive pressure recovery.

All put together, the pressure profiles Eta was designed for boast a linear laminar
flow region combined with a Stratford pressure recovery. [4]
Figure 1.1: Eta

Figure 1.2: Eta Inside
Figure 1.3: Eta top View
1.2 Drag Coefficient and Reynolds Number

Plotting drag coefficient of an axisymmetric body for Reynolds number’s range, we can notice three different regions. For Reynolds number lower than $5 \cdot 10^6$ is possible to have extensive laminar flow thanks to a low skin friction. Increasing Reynolds number, transition points move towards the body nose increasing drag coefficient.

![Figure 1.4: Drag Coefficient versus Reynolds Number](image)

In Reynolds $> 10^7$ region boundary layer is almost fully turbulent. An important task for aerodynamics is to determine how to shape the body geometry to delay transition and to realize extensive laminar flow. However, it is not known to what extent the theoretically evaluated laminar flow can be realized with actual airship applications with a certain degree of surface waviness. For the shape optimizations presented in this work, it became obvious
that one-point optimizations for a single Reynolds number lead to bodies which are inconvenient or even unusable outside of their design point. This is especially true for laminar bodies at low Reynolds numbers. Following is reported shaping optimization at various Reynolds Numbers.[5]

- Regime I: $1 \cdot 10^6 < Re < 3.16 \cdot 10^6$
- Regime II: $3.16 \cdot 10^6 < Re < 1 \cdot 10^7$
- Regime III: $1 \cdot 10^7 < Re < 3.16 \cdot 10^7$
- Regime IV: $3.16 \cdot 10^7 < Re < 1 \cdot 10^8$

The initial source distribution chosen for design regime I corresponded to an ellipsoid-like starting geometry with a small length-to-diameter ratio of $L/D=2.3$.

Figure 1.5: Shaping Optimization versus Reynolds Number
When comparing body contour and pressure distribution, it can be noted that minimum pressure coefficient occurs downstream of the maximum thickness point. Increasing Reynolds number, in order to maintain laminar flow, the amount of favorable pressure gradient in the forebody region has to be enlarged. This can be realized in two different ways:

- increasing the body diameter
- moving the maximum thickness point upstream
Chapter 2

Physics problem

2.1 Aerodynamic Drag

Aerodynamic drag is the fluid drag force that acts on any moving solid body in the direction of the fluid freestream flow. From the body’s perspective (near-field approach), the drag results from forces due to pressure distributions over the body surface. The surrounding fluid exerts pressure forces and viscous forces on an object. The components of the resultant force acting on the object are the drag force and the lift force and both are influenced by the size and shape of the object and the Reynolds number of the flow.

\[
F_D = \frac{1}{2} \rho U^2 C_D A \\
R_e = \frac{\rho U D}{\nu}
\]

(2.1)
(2.2)

Where:

- \(F_D\): aerodynamic force
- \(\rho\): fluid density
- \(U\): body’s speed
- \(c_D\): aerodynamic coefficient
2 – Physics problem

- \( A \): cross sectional area
- \( \text{Re} \): Reynolds Number
- \( \nu \): dynamic viscosity.

The drag force is due to the pressure and shear forces acting on the surface of the object.

\[
F_{D, \text{viscous}} = A \cdot \tau_w \quad (2.3)
\]

\[
F_{D, \text{pressure}} = \int_A p \cdot da_n \quad (2.4)
\]
2.2 Laminar and Turbulent Flow

The primary characteristic of laminar flow is a streamlined flow, lacking any swirls or cross currents. If one imagines different layers of a fluid, divided into rows/cylinders at various radii, the layers of the fluid wouldn’t mix. The fluid would flow without interference or disturbance, and the path of the flow wouldn’t have any swirls or cross currents.

The layers or “tubes” of the fluid would still flow at different speeds, even though they wouldn’t cross or intersect. The central, innermost layer would have the fastest flow speed while the outer layers would have a much slower flow speed (sometimes hardly moving at all). In general, the velocity of a fluid with laminar flow is extremely low. In fluid dynamics, laminar flow occurs when a fluid flows in parallel layers, with no breaks between the layers. The fluid tends to flow without lateral mixing, and adjacent layers slide past one another like playing cards. There are no cross-currents perpendicular to the direction of flow, nor eddies or swirls of fluids. In laminar flow, the motion of the particles of the fluid is very orderly with particles close to a solid surface moving in straight lines parallel to that surface. Laminar flow is a flow regime characterized by high momentum diffusion and low momentum convection.

Laminar Flow could be described as the flow of a fluid whenever each and every particle belonging to the fluid is a follower of a consistent course, routes which usually under no circumstances obstruct with one another. One consequence of laminar movement would be that the speed belonging to the fluid is actually constant at any time inside fluid whereas on the other hand Turbulent Flow could be described as the uneven, unfrequented movement of fluid which is seen as a small whirlpool
areas. The speed of such a fluid is unquestionably not necessarily constant at each and every point. Turbulent flow is characterized by the chaotic and rough movement of particles through a region. Imagining layers of a fluid again, you can think of the various layers mixing with one another, with a heavy amount of friction existing between the boundaries of the different layers. Molecules are thrown around in an irregular fashion, and whirlpools and waves can easily be found within the flow. Fluids that are in turbulent flow have a substantial amount of kinetic energy within them. As long as this energy persists, the flow will continue to be turbulent and irregular. Once the energy is used up, the flow transitions to a laminar state. The velocity of a turbulent fluid is typically high, equalizing as it transitions to a laminar flow state.

When a fluid is flowing through a closed channel such as a pipe or between two flat plates, either of two types of flow may occur depending on the velocity and viscosity of the fluid: laminar flow or turbulent flow. Laminar flow tends to occur at lower velocities, below a threshold at which it becomes turbulent. Turbulent flow is a less orderly flow regime that is characterised by eddies or small packets of fluid particles, which result in lateral mixing. In non-scientific terms, laminar flow is smooth, while turbulent flow is rough.

Fluids frequently do transition between turbulent and laminar flows. This transition between the two different kinds of flows is impacted by a variety of different variables. Changes in the flow of a fluid may be driven by interactions with an object moving through the air, causing layers of a fluid to mix or straighten out as it moves along. For instance, while the air that moves over the wing of a plane is generally flowing in a laminar fashion, the air surrounding the plain is probably flowing turbulently. The tips of airplane wings often create a tip vortex, which causes the air in that region to begin flowing turbulently.
2.3 Boundary Layers and Separation

Now the body is invested by the flow. According to slip condition, the fluid velocity on surface body is zero; the layers closer to the wall start moving right away due to the no-slip boundary condition and later start moving the layers farther away from the wall. The distance from the wall that is affected by the motion is also called viscous diffusion length. The layers close to the wall are dragged along while the layers farther away from the wall move with a lower velocity. The viscous layer develops as a result of the no-slip boundary condition at the wall. [7]

When viscous layer velocity reaches the 99% of undisturbed speed the viscous layer ends.

![Velocity Profile](image)

Figure 2.3: Velocity Profile

Exact equations for velocity profile in viscous boundary layer were derived by Stokes in 1881.

From the Navier-Stokes equation:

\[
\frac{\delta u}{\delta t} = \nu \frac{\delta^2 u}{\delta t^2}
\]

(2.5)

Derive exact solution for the velocity profile:

\[
U = U_0 \left(1 - erf \left(\frac{y}{2\sqrt{\nu t}}\right)\right)
\]

(2.6)

where erf is the error function.
The boundary layer thickness can be approximated by.

\[ \frac{\delta u}{\delta t} = \nu \frac{\delta^2 u}{\delta t^2} \rightarrow \frac{U_0}{t} \approx \nu \frac{U_0}{\delta^2} \rightarrow \delta \approx \sqrt{\nu t} \]  \hspace{1cm} (2.7)

The flow accelerate in the convergence zone; as flow goes faster, it reduces it pressure. Flow reach maximum speed and minimum pressure at through (minimum section). In the divergence zone he starts to decelerate to enviroment conditions. Flow goes from a lower pressure zone to an higher. Pressure gradient will be trying to slow the flow down which is exactly what we’re seeing, the problem is that if the adverse pressure gradient is too high it will slow the boundary layer too much, the pressure gradient have enough force forcing the flow back that way that the boundary layer will try to return back, this causes our velocity look like (S3), which is essentially a recirculation region which is stall and this is caused by the adverse pressure gradient that pushing boundary layer flow backwards.

Figure 2.4: Convergence-Divergence Nozzle

Figure 2.5: Separation Point
2.3.1 Transition Uncertainty

The prediction of boundary layer transition at assigned speeds becomes a complicated task for the aerodynamicist since the phenomenon is affected by a very large number of parameters which are difficult to determine or evaluate.[10] Indeed although the flow in the convergence zone should be laminar, the manufacturing process can affect a premature transition of the boundary layer from laminar to turbulent flow. It can be produced by the local increases of the Reynolds number. In practice how the roughness of the surface influences on the laminar to turbulent transition isn’t easy to determinate even if roughness is ”a priori” known. Boundary layer transition depends on many coupled parameters. On design transition should usually occur symmetrically on both left and right sides of the vehicle, but the many parameters that influences it can cause premature transition. Laminar-turbulent transition unites two types of uncertainty. The first one is a random uncertainty (”the stochastic uncertainty”) that is associated with inherent variations in the physical system or its environment. The body’s smooth windward surface after repeated use or shots, might eventually become a continous pattern of asperities that could lead a premature transition to turbulent flow. The second type of uncertainty is the result of the lack of knowledge that arises from the use of inadequate physical models. Since there is no universal value for the transitional Reynolds number ReT, the inherent uncertainty in predictions that results from the evaluation of this parameter, for instance through correlations such as 2.8, needs to be be quantified in order to obtain a measure of the robustness of the design process.

\[
\log_{10}(Re_T) = 6.421 \exp(1.209 \cdot 10^{-4} M_{edge}^{2.641})
\]  

(2.8)
Chapter 3

STAR-CCM+

STAR-CCM+ (Simulation of Turbulent flow in Arbitrary Regions - Computational Continuum Mechanics) is a software for computational fluid dynamics analysis (CFD), based on finite element method (FEM).

3.1 Finite Element Method

The Finite Element Method (FEM) is a numerical method based on the integration of partial differential equations on a volume control in which are established boundary solution. The total domain is divided into a collection of elementary volumes, the differential equation are calculated on any of this elementary volumes resulting in a system of algebraic equations that are be solved by calculator. The subdivision of a whole domain into simpler parts has several advantages: a better accurate representation of complex geometry, an easy representation of the total solution and the possibility to capture of local effects.

A simple cylinder body with a bad mesh loses its property. Analyzing a bad meshes body there will be unphysics solutions. The simulator software runs anyway but it is in the user’s ability to analyze results and validate them.
Figure 3.1: Geometry Body - Bad Mesh - Good Mesh
3.2 Geometry

3.2.1 CAD-Models: B-2019

A solid of revolution is obtained by rotating a plane figure in space about an axis coplanar to the figure. Axissymmetric body means that almost two of its three principal moments of inertia are equal. The starting body used in this thesis (B-2019) is characterized by 8 parameters, increased to 10 improving mesh representation. The 10 parameters are:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>[m]</th>
<th>Fixed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pedals Position</td>
<td>PP</td>
<td>0.64</td>
<td>✓</td>
</tr>
<tr>
<td>Maximum Diameter Position</td>
<td>MDP</td>
<td>1.1</td>
<td>□</td>
</tr>
<tr>
<td>Shoulders Position</td>
<td>SP</td>
<td>1.74</td>
<td>✓</td>
</tr>
<tr>
<td>Inflection Position</td>
<td>IP</td>
<td>1.85</td>
<td>□</td>
</tr>
<tr>
<td>Cut Position</td>
<td>CP</td>
<td>2.55</td>
<td>□</td>
</tr>
<tr>
<td>Total Length</td>
<td>L</td>
<td>2.60</td>
<td>□</td>
</tr>
<tr>
<td>Pedals Diameter</td>
<td>PD</td>
<td>0.227</td>
<td>✓</td>
</tr>
<tr>
<td>Maximum Diameter</td>
<td>MD</td>
<td>0.25</td>
<td>□</td>
</tr>
<tr>
<td>Shoulders Diameter</td>
<td>SD</td>
<td>0.18</td>
<td>✓</td>
</tr>
<tr>
<td>Inflection Diameter</td>
<td>ID</td>
<td>0.15</td>
<td>□</td>
</tr>
</tbody>
</table>

Table 3.1: B-2019 Parameters

Values are taken from Tarurus CAD Model. In order to guarantee necessary cyclist’s comfort, are fixed three parameters: Pedals & Shoulders Positions, Pedals Diameters and Shoulders Diameter. In particular, during optimization, pedals position and shoulders one are not fixed separately, they are related by their distance, PP and SP could translate but their distance will be unchanged (1.1 m). The same speech goes for total length: 5 cm far away from CP.
Figure 3.2: B-2019 Front and Top Views
3.2.2 Gallery

CAD-Model is located in a 3D solid that acts like a wind gallery.

![Wind Gallery Image](image)

Figure 3.3: Wind Gallery

Wind gallery is big enough to let a flow natural motion. Through Boolean operation B-2019 is subtract to the gallery. In this way the simulator software analyze the total volume whitout any solid part, since we are non interested to what happen inside the nondeformable body.

<table>
<thead>
<tr>
<th>Axis</th>
<th>[m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>30</td>
</tr>
<tr>
<td>Y</td>
<td>8</td>
</tr>
<tr>
<td>Z</td>
<td>4</td>
</tr>
</tbody>
</table>

Table 3.2: Gallery Dimensions
3.3 Physics Condition

3.3.1 All \( y^+ \) Wall Treatment

Walls are a source of vorticity in most flow problems of practical importance. Therefore, an accurate prediction of flow and turbulence parameters across the wall boundary layer is essential. The inner region of the boundary layer can be split up into three sublayers. In each of them the flow has different characteristics and can be modeled using different empirical approaches:

- **Viscous sublayer:** The fluid layer in contact with the wall is dominated by viscous effects and is almost laminar. The mean flow velocity only depends on the fluid density, viscosity, distance from the wall, and the wall shear stress.

- **Log-law layer:** The turbulent log-law layer is dominated equally by viscous and turbulent effects.

- **Buffer layer:** The buffer layer is a transitional layer between the viscous sublayer and the log-law layer.

The non-dimensional wall distance can be used to define the extents of the sublayers. The following plot shows the non-dimensional velocity as a function of across the three sublayers:

![Non-dimensional velocity as a function of across the three sublayers](image)

Figure 3.4: Non-dimensional velocity as a function of across the three sublayers
The low-y+ wall treatment resolves the viscous sublayer and needs little or no modeling to predict the flow across the wall boundary. The transport equations are solved all the way to the wall cell. The wall shear stress is computed as in laminar flows. To resolve the viscous sublayer, these models require a sufficiently fine mesh with near-wall cells located at or around unity. The computational expense that is associated with this approach can be significant, particularly for large Reynolds number flows where the viscous sublayer can be very thin. Therefore this wall treatment is suitable only for low Reynolds number flows.[8]

Figure 3.5: low-y+ approach
3.3.2 Gamma Re-Theta Transition Model

The Gamma Re-Theta Transition model is a two-equation correlation-based transition model that provides a true predictive capability for the onset of transition in a turbulent boundary layer. Gamma-ReTheta Transition model allows you to solve for the turbulence intermittency $\gamma$ and the transition momentum thickness Reynolds number $Re_\theta$ to predict the onset of transition in a turbulent boundary layer.

The evaluation of momentum thickness Reynolds number is avoided by relating this quantity to vorticity-based Reynolds number. In addition, a correlation for transition onset momentum thickness Reynolds number defined in the free stream is propagated into the boundary layer by a transport equation. An intermittency transport equation is further used in such a way that the source terms attempt to mimic the behavior of algebraic engineering correlations.

The Gamma ReTheta transition model is incomplete, since two critical correlations were claimed to be proprietary and hence omitted. Therefore, the model’s implementation is done in such a way as to allow the custom specification of correlations via field functions. Recognizing, however, that the process of calibrating the needed correlations is complex and time consuming, the required correlations have been carefully calibrated and are provided by default within the model.

| Convection | 2-nd order
| Cross flow term | □
| Free Stream Edge | TBD

Table 3.3: Gamma ReTheta Parameters
3.3.3 Free Stream Edge

Gamma ReTheta Transition Model needs a free stream edge function that better describes the evolution of viscous boundary layer. This free-stream value is transported into the boundary layer through the transition momentum thickness Reynolds number $Re_\theta$. In order to specify the value of $Re_\theta$ using a correlation, the location of the free-stream edge must be defined. Since there is no completely general way to define the free stream, it is up to the user to create a user field function that suitably defines the free stream in such a way that the field function takes the value of 1 in the free stream and 0 inside the boundary layer. A good starting point for this definition is in terms of Wall Distance.

An example would be:

$$WallDistance > 0.005$$  \hspace{1cm} (3.1)

where it is estimated that the boundary layer is everywhere thinner than 5 mm. The user field function, once defined and named, must then be assigned to the Free Stream Edge property of the Gamma ReTheta transition model.[8] In order to formulate the function that STAR-CCM+ uses, has been analyzed a 13 inches specimen.
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0.9</td>
<td>0.5293</td>
<td>5.8</td>
<td>1.3164</td>
<td>9.45</td>
<td>0.7346</td>
</tr>
<tr>
<td>0.001</td>
<td>0.0163</td>
<td>1</td>
<td>0.5614</td>
<td>6</td>
<td>1.3243</td>
<td>9.5</td>
<td>0.712</td>
</tr>
<tr>
<td>0.002</td>
<td>0.0231</td>
<td>1.2</td>
<td>0.6218</td>
<td>6.2</td>
<td>1.3284</td>
<td>9.55</td>
<td>0.6894</td>
</tr>
<tr>
<td>0.004</td>
<td>0.0327</td>
<td>1.4</td>
<td>0.6781</td>
<td>6.4</td>
<td>1.3317</td>
<td>9.6</td>
<td>0.6667</td>
</tr>
<tr>
<td>0.006</td>
<td>0.04</td>
<td>1.6</td>
<td>0.7308</td>
<td>6.6</td>
<td>1.3332</td>
<td>9.65</td>
<td>0.644</td>
</tr>
<tr>
<td>0.008</td>
<td>0.0462</td>
<td>1.8</td>
<td>0.7805</td>
<td>6.666</td>
<td>1.3333</td>
<td>9.7</td>
<td>0.6215</td>
</tr>
<tr>
<td>0.01</td>
<td>0.0517</td>
<td>2</td>
<td>0.8274</td>
<td>6.8</td>
<td>1.3329</td>
<td>9.75</td>
<td>0.5994</td>
</tr>
<tr>
<td>0.015</td>
<td>0.0634</td>
<td>2.2</td>
<td>0.8717</td>
<td>7</td>
<td>1.3297</td>
<td>9.8</td>
<td>0.5778</td>
</tr>
<tr>
<td>0.02</td>
<td>0.0732</td>
<td>2.4</td>
<td>0.9136</td>
<td>7.2</td>
<td>1.3225</td>
<td>9.85</td>
<td>0.5569</td>
</tr>
<tr>
<td>0.025</td>
<td>0.0819</td>
<td>2.6</td>
<td>0.9532</td>
<td>7.4</td>
<td>1.31</td>
<td>9.9</td>
<td>0.5368</td>
</tr>
<tr>
<td>0.03</td>
<td>0.0898</td>
<td>2.8</td>
<td>0.9907</td>
<td>7.6</td>
<td>1.291</td>
<td>10</td>
<td>0.4994</td>
</tr>
<tr>
<td>0.04</td>
<td>0.1037</td>
<td>3</td>
<td>1.0259</td>
<td>7.8</td>
<td>1.2645</td>
<td>10.1</td>
<td>0.4663</td>
</tr>
<tr>
<td>0.05</td>
<td>0.1161</td>
<td>3.2</td>
<td>1.0592</td>
<td>8</td>
<td>1.23</td>
<td>10.2</td>
<td>0.4378</td>
</tr>
<tr>
<td>0.06</td>
<td>0.1273</td>
<td>3.4</td>
<td>1.0904</td>
<td>8.2</td>
<td>1.1868</td>
<td>10.3</td>
<td>0.4141</td>
</tr>
<tr>
<td>0.07</td>
<td>0.1377</td>
<td>3.6</td>
<td>1.1196</td>
<td>8.4</td>
<td>1.1348</td>
<td>10.4</td>
<td>0.3953</td>
</tr>
<tr>
<td>0.08</td>
<td>0.1474</td>
<td>3.8</td>
<td>1.1468</td>
<td>8.5</td>
<td>1.1055</td>
<td>10.5</td>
<td>0.3811</td>
</tr>
<tr>
<td>0.09</td>
<td>0.1565</td>
<td>4</td>
<td>1.1722</td>
<td>8.6</td>
<td>1.074</td>
<td>10.6</td>
<td>0.3713</td>
</tr>
<tr>
<td>0.1</td>
<td>0.1651</td>
<td>4.2</td>
<td>1.1956</td>
<td>8.7</td>
<td>1.0404</td>
<td>10.7</td>
<td>0.3654</td>
</tr>
<tr>
<td>0.2</td>
<td>0.2359</td>
<td>4.4</td>
<td>1.2172</td>
<td>8.8</td>
<td>1.0048</td>
<td>10.8</td>
<td>0.3629</td>
</tr>
<tr>
<td>0.3</td>
<td>0.2917</td>
<td>4.6</td>
<td>1.2369</td>
<td>8.9</td>
<td>0.9673</td>
<td>11</td>
<td>0.3659</td>
</tr>
<tr>
<td>0.4</td>
<td>0.3399</td>
<td>4.8</td>
<td>1.2547</td>
<td>9</td>
<td>0.928</td>
<td>11.2</td>
<td>0.3752</td>
</tr>
<tr>
<td>0.5</td>
<td>0.3832</td>
<td>5</td>
<td>1.2707</td>
<td>9.1</td>
<td>0.8872</td>
<td>11.4</td>
<td>0.3861</td>
</tr>
<tr>
<td>0.6</td>
<td>0.4232</td>
<td>5.2</td>
<td>1.2849</td>
<td>9.2</td>
<td>0.8449</td>
<td>11</td>
<td>0.3947</td>
</tr>
<tr>
<td>0.7</td>
<td>0.4605</td>
<td>5.4</td>
<td>1.2972</td>
<td>9.3</td>
<td>0.8015</td>
<td>11.8</td>
<td>0.3992</td>
</tr>
<tr>
<td>0.8</td>
<td>0.4958</td>
<td>5.6</td>
<td>1.3077</td>
<td>9.4</td>
<td>0.7571</td>
<td>12</td>
<td>0.4</td>
</tr>
</tbody>
</table>

Table 3.4: 13-Inches Profile Plot Data
The field function that better revive 13 inches specimen experimental results is:

\[
\frac{V^2}{2 \cdot \left( \frac{p_{ref} - p_0}{\rho_{ref}} \right) + S^2} > 0.9801\|W_D > W_{D_{max}} \tag{3.2}
\]

Where:

- \( V \) (Velocity) is the elementar cell local speed
- \( S \) (Speed) is the reference speed (150 kph)
- \( \rho=\rho_{ref} \) since constant density model
- \( p_{ref} \) is the reference pressure (101 325 Pa)
- \( W_D \) is the elementar cell distance from the wall

The function study boundary layer from Bernoulli’s principle.

\[
\frac{1}{2} \rho V^2 + p = cost \tag{3.3}
\]

Specialized for viscous region it becomes:

\[
\frac{1}{2} \rho V^2 + p = \frac{1}{2} \rho_{ref} (99\% \cdot S)^2 + p_{ref} \tag{3.4}
\]
According to experimental results, Star-CCM+ Drag Coefficient and Experimental drag coefficient are above 0.04. Also, through Wall Shear Stress analysis, it could be possible estimated transition point. Both Experimental and Star-CCM+ points are above the 60% of the total length. [9]

Analyzing the Field Function is possible to notice its trend. It starts from zeros, or little more, at the nose till reach 16 mm at the profile ending point. This assignment is physically correct since field function isn’t constant across the body like starting function allowed to understand.
3.3.4 Wall Shear Stress

Figure 3.9 represents the Wall Shear Stress ($\tau_w$) on 13-inches specimen.

It is the component of stress coplanar with a material cross section. Shear stress arises from the force vector component parallel to the cross section of the material. Any real fluids moving along a solid boundary will incur a shear stress at that boundary. The no-slip condition dictates that the speed of the fluid at the boundary (relative to the boundary) is zero; although at some height from the boundary the flow speed must equal that of the fluid. For all Newtonian fluids in laminar flow, the shear stress is proportional to the strain rate in the fluid, where the viscosity is the constant of proportionality. The shear stress is imparted onto the boundary as a result of this loss of velocity.

For a Newtonian fluid, the shear stress at a surface element parallel to a flat
plate at the point \( y \) is given by:

\[
\tau(y) = \frac{\partial u}{\partial y} \tau(y) = \frac{\partial u}{\partial y}
\]  

(3.5)

Where:

- \( \mu \) is the dynamic viscosity of the flow;
- \( u \) is the flow velocity along the boundary;
- \( y \) is the height above the boundary.

Specifically, the Wall Shear Stress is given by:

\[
\tau_w = \mu \left( \frac{\delta u}{\delta y} \right)_{y=0}
\]  

(3.6)

It plays a significant role in transition point’s analysis. When its value starts to become higher, it means that laminar to turbulent transition process starts, viscous boundary layer becomes more unstable until the detachment from the body.

The SI unit of wall shear stress is pascal (Pa), which is identical to \( \frac{kg}{m \cdot s^2} \).
3.3.5 Cell Quality Remediation

The Cell Quality Remediation model helps you get solutions on a poor-quality mesh. This model identifies poor-quality cells, using a set of predefined criteria, such as Skewness Angle exceeding a certain threshold. Once these cells and their neighbors have been marked, the computed gradients in these cells are modified in such a way as to improve the robustness of the solution. In general, the effect of Cell Quality Remediation is confined to the immediate vicinity of poor-quality and/or degenerate cells, so that the influence on overall solution accuracy is minimal.[8]

3.3.6 Exact Wall Distance

The Exact Wall Distance model makes an exact projection calculation in real space, which is based on a triangulation of the surface mesh. The use of K-D search trees or SFC/SIMD algorithms accelerates the calculation. The Exact Wall Distance model is selected on your behalf after the selection of a specific physical model that requires the wall distance parameter.[8]

3.3.7 Gas

Gas used is Air in standard conditions.

<table>
<thead>
<tr>
<th></th>
<th>1.225 kg/m³</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>1.8 × 10⁻⁵ Pa · s</td>
</tr>
<tr>
<td>Dynamic Viscosity</td>
<td>150 kph</td>
</tr>
<tr>
<td>Speed</td>
<td>101 325 Pa</td>
</tr>
</tbody>
</table>

Table 3.5: Air Data
3.3.8 Turbulent

A flow that is in a state of continuous instability, exhibiting irregular, small-scale, high-frequency fluctuations in both space and time is termed turbulent. It is strictly possible to simulate turbulent flow directly by resolving all the scales of the flow (termed direct numerical simulation). However, the computer resources that are required are too large for practical flow simulations. Therefore, a suitable turbulence modeling approach must be selected.[8]

3.3.9 Segregated Flow

The Segregated Flow model solves the flow equations (one for each component of velocity, and one for pressure) in a segregated, or uncoupled, manner. The linkage between the momentum and continuity equations is achieved with a predictor-corrector approach.[8]

Other enabled models:

- Constant Density
- Steady
- Three Dimensional Model
3.4 Regions

Regions are volume domains (or areas in a two-dimensional case) in space that are completely surrounded by boundaries. They are discretized by a conformal mesh consisting of connected faces, cells and vertices. One region is joined to another using an interface, so that information can be passed between them. Boundaries are not shared between regions. There may be multiple regions in a simulation, each with a unique name and properties.

Region nodes are created when you:

- Import a mesh, ad exemple from native CAD
- Split a non-contiguous region
- Assign parts to regions

Creating regions from existing boundaries is possible to apply different physical properties to a portion of the fluid domain. To each region are assigned:

- Mesh Continuum
- Physics Continuum
- Parts
- Type
In the studied case there is one region and four difference boundaries:

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
<th>Shear Stress Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>(A) Inlet</td>
<td>Velocity Inlet</td>
<td>n/a</td>
</tr>
<tr>
<td>(B) Outlet</td>
<td>Pressure Outlet</td>
<td>n/a</td>
</tr>
<tr>
<td>(C) Body</td>
<td>Wall</td>
<td>No-slip</td>
</tr>
<tr>
<td>(D) Ground</td>
<td>Wall</td>
<td>Slip</td>
</tr>
</tbody>
</table>

Table 3.6: Boundaries
3.5 Mesh Conditions

The total volume is divided into 4161718 trimmed elementary cells that better describes the geometry and physics problems. The base size for the surface mesh is 3 cm, while the volume mesh target is the 0.1% (0.01 m). There are 16 prism layers for 16 mm and the nearest layer is far 0.16 μm from the body wall. Obviously the mesh is thicker near the wall and grows away from it. Beyond the body there is a 8 meters region (across X) that analyzes the trail.
### Table 3.7: Mesh Conditions

<table>
<thead>
<tr>
<th>Controls</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cells</td>
<td>4161718</td>
</tr>
<tr>
<td>Base Size</td>
<td>10 m</td>
</tr>
<tr>
<td>Target Surface Size</td>
<td>0.006 m</td>
</tr>
<tr>
<td>Minimum Surface Size</td>
<td>0.003 m</td>
</tr>
<tr>
<td>Surface Growth Rate</td>
<td>2.0</td>
</tr>
<tr>
<td>Auto-Repair Minimum Proximity</td>
<td>$1.0 \times 10^{-4}$ m</td>
</tr>
<tr>
<td>Prism Layer Near Wall Thickness</td>
<td>$1.6 \times 10^{-5}$ m</td>
</tr>
<tr>
<td>Number of Prism Layers</td>
<td>16</td>
</tr>
<tr>
<td>Prism Layer Total Thickness</td>
<td>0.016 m</td>
</tr>
<tr>
<td>Default Growth Rate</td>
<td>Fast</td>
</tr>
<tr>
<td>Surface Growth Rate</td>
<td>Very Slow</td>
</tr>
<tr>
<td>Distance</td>
<td>8 m</td>
</tr>
<tr>
<td>Wake Refinement</td>
<td>0.1 m</td>
</tr>
</tbody>
</table>
3.6 Plots and Results

Running Star-CCM+ for 1500 iterations, in order to have stationarity and convergence, the results are:

Figure 3.14: Residuals

Figure 3.15: Drag
There is convergence in results since errors are settled on constant and very little values. Green curve represents Gamma Re-Theta Transition Model, that is the last residual to converge (1000 iterations).
Drag value is of 6.12 N.

Thanks to WSS scene is possible to analyze transition. It is located after the maximum diameter point, at almost the 60% of the total length.
Speed and pressure values are in accordance to physics problem. In correspondence of profile leading edge and the ending edge, where there are two stopping points, we have maximum pressure and zero velocity. Velocity increases till its maximum (and minimum pressure) point that is located in correspondence of B-2019 maximum diameter.
The total pressure doesn’t change except for numerical errors that are brought to valley by the viscousity introducted by the numerical method. The viscosity effect causes the damping of the solutions, a small results’ precision lost but makes the numerical method stable.
Chapter 4

**HEEDS**

The shape optimization problem is solved using HEEDS software. HEEDS allows the user to compare difference simulation results modifying design parameters.

### 4.0.1 Parameters

There are 2 different classes of parameters: variables and responses. The first ones could change their values in order to get different results; the second parameters are the mission target, what HEEDS research to accomplish the mission. The only target is to minimize the Drag Force value. Fixing pedals and shoulder’s diameters and their distance, the other eight parameters could be combined as follows:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>min [m]</th>
<th>Baseline [m]</th>
<th>max [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>PP</td>
<td>Continuos</td>
<td>0.4</td>
<td>0.64</td>
<td>0.8</td>
</tr>
<tr>
<td>MDP</td>
<td>Continuos</td>
<td>0.9</td>
<td>1.1</td>
<td>1.5</td>
</tr>
<tr>
<td>SP</td>
<td>Dependent</td>
<td></td>
<td>PP+1.1</td>
<td></td>
</tr>
<tr>
<td>IP</td>
<td>Continuos</td>
<td>1.7</td>
<td>1.85</td>
<td>2</td>
</tr>
<tr>
<td>CP</td>
<td>Continuos</td>
<td>2.4</td>
<td>2.55</td>
<td>2.8</td>
</tr>
<tr>
<td>L</td>
<td>Dependent</td>
<td></td>
<td>CP+0.05</td>
<td></td>
</tr>
<tr>
<td>MD</td>
<td>Continuos</td>
<td>0.23</td>
<td>0.26</td>
<td>0.4</td>
</tr>
<tr>
<td>ID</td>
<td>Continuos</td>
<td>0.1</td>
<td>0.15</td>
<td>0.2</td>
</tr>
<tr>
<td>PD</td>
<td>Fixed</td>
<td></td>
<td>0.227</td>
<td></td>
</tr>
<tr>
<td>SD</td>
<td>Fixed</td>
<td></td>
<td>0.18</td>
<td></td>
</tr>
</tbody>
</table>

*Table 4.1: HEEDS Parameters*
4.1 Results

After analyzing over 100 parameters combinations, HEEDS established that best configurations are:

4.1.1 Design A

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>PP</td>
<td>0.64</td>
</tr>
<tr>
<td>MDP</td>
<td>1.1</td>
</tr>
<tr>
<td>SP</td>
<td>1.74</td>
</tr>
<tr>
<td>IP</td>
<td>1.81</td>
</tr>
<tr>
<td>CP</td>
<td>2.52</td>
</tr>
<tr>
<td>L</td>
<td>2.57</td>
</tr>
<tr>
<td>MD</td>
<td>0.26</td>
</tr>
<tr>
<td>ID</td>
<td>0.16</td>
</tr>
<tr>
<td>PD</td>
<td>0.227</td>
</tr>
<tr>
<td>SD</td>
<td>0.18</td>
</tr>
</tbody>
</table>

Table 4.2: Design A - Parameters

The first design has a Drag Force value similar to the original shape. This design has inflection position 4 cm forward the original design; it is 3 cm shorter and has the inflection diameter higher than B-2019. All this variation lead to a bigger curvature in order to manintain the zero grades tangency at the ending edge. Star analysis show no differences in drag force.
Figure 4.2: Design A Residuals

Figure 4.3: Design A Drag

Figure 4.4: Design A Wall Shear Stress
4.1.2 Design B

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>PP</td>
<td>0.64</td>
</tr>
<tr>
<td>MDP</td>
<td>1.1</td>
</tr>
<tr>
<td>SP</td>
<td>1.74</td>
</tr>
<tr>
<td>IP</td>
<td>1.85</td>
</tr>
<tr>
<td>CP</td>
<td>2.53</td>
</tr>
<tr>
<td>L</td>
<td>2.58</td>
</tr>
<tr>
<td>MD</td>
<td>0.26</td>
</tr>
<tr>
<td>ID</td>
<td>0.16</td>
</tr>
<tr>
<td>PD</td>
<td>0.227</td>
</tr>
<tr>
<td>SD</td>
<td>0.18</td>
</tr>
</tbody>
</table>

Table 4.3: Design B - Parameters

Thanks this new shape drag is reduced of 1%.
There is convergence in results since residuals do not change their values during iterations; transiction point is located after maximum diameter, 1.1 m from the nose. In this way laminar flow could reach over the 60% of the total lenght.
There’s not differences in speed and pressure evolution (convergence-diveregence nozzle).
Total pressure show the usual machine error, due to truncation, and the viscosity introduced by numerical method.
A significan little difference with other shapes is the fast tangency change in the queue due to the fact that B-Design is 2 cm shorter than the original shape (keeping the other parameters unchanged).
Figure 4.6: Design B Residuals

Figure 4.7: Design B Drag

Figure 4.8: Design B Wall Shear Stress
Figure 4.9: Design B Speed and Pressure

Figure 4.10: Design B Total Pressure
4.1.3 Design C

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>PP</td>
<td>0.64</td>
</tr>
<tr>
<td>MDP</td>
<td>1.1</td>
</tr>
<tr>
<td>SP</td>
<td>1.78</td>
</tr>
<tr>
<td>IP</td>
<td>1.81</td>
</tr>
<tr>
<td>CP</td>
<td>2.52</td>
</tr>
<tr>
<td>L</td>
<td>2.57</td>
</tr>
<tr>
<td>MD</td>
<td>0.27</td>
</tr>
<tr>
<td>ID</td>
<td>0.16</td>
</tr>
<tr>
<td>PD</td>
<td>0.227</td>
</tr>
<tr>
<td>SD</td>
<td>0.18</td>
</tr>
<tr>
<td>Drag</td>
<td>5.94 N</td>
</tr>
</tbody>
</table>

Table 4.4: Design C - Parameters

C shape is the best Design.

In this configuration Maximum Diameter is increased by 2 mm. This could lead to a sudden detachment of laminar flow but, increasing Inflection Diameter and its approach to the shoulder position (4 mm nearest), speed flow increases softly and laminar flow grows more in the divergence zone, a determinant variation for the reduction of aerodynamic drag.

It is 3 cm shorter than B-2019 Design.

Speed, pressure and total pressure doesn’t show any particular differences from other solutions, while the total pressure show the usual viscosity effect.
Figure 4.12: Design C Drag and Residuals

Figure 4.13: Design C WSS and Wally+
Figure 4.14: Design C Speed and Pressure

Figure 4.15: Design C Total Pressure
Even there is a little difference in the input parameters, HEEDS result gives better performance and an almost 3% drag decrease.

<table>
<thead>
<tr>
<th>Body</th>
<th>Drag [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>B-2019</td>
<td>6.12</td>
</tr>
<tr>
<td>Design A</td>
<td>6.12</td>
</tr>
<tr>
<td>Design B</td>
<td>6.06</td>
</tr>
<tr>
<td>Design C</td>
<td>5.94</td>
</tr>
</tbody>
</table>

Table 4.5: Design Comparison

C-Design is the best shape at given Reynolds and speed. In the following study we consider this best design as the default one.
Chapter 5

Ground Effect

In the previous chapter B-2019 study was carried out in air, the body has been placed at a distance from the boundaries that let the normal trend flow. In this chapter B-2019 will be near to the ground in order to consider Ground Effect. In computational analysis term it means to apply no-slip condition to the wall, the flow adheres to it. Viscous boundary layer must be considered also on the ground; there flow speed is zero, it grows away from the wall till the 99% where boundary layer ends. In order to describe better this phenomenon will be added, to the original mesh, 16 prism layers to the ground.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
<th>Shear Stress Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>(A) Inlet</td>
<td>Velocity Inlet</td>
<td>n/a</td>
</tr>
<tr>
<td>(B) Outlet</td>
<td>Pressure Outlet</td>
<td>n/a</td>
</tr>
<tr>
<td>(C) Body</td>
<td>Wall</td>
<td>No-slip</td>
</tr>
<tr>
<td>(D) Ground</td>
<td>Wall</td>
<td>No-slip</td>
</tr>
</tbody>
</table>

Table 5.1: Region Boundaries Condition

There are two modes to analyze it. In the first mode B-2019 original shape is slightly modified with addition of two "wheels", in the second mode the wheels were removed since they have a less important role in drag force computation.
5.0.1 With Wheels

As previously said body shape is modified. To the original body have been added two cylinders, that represents the two wheels. Volume mesh count 4761321 cells since it has improved in correspondence of the cylinders and the ground. Adding two cylinders we lost the axissymmetric geometry shape of the original body.

Figure 5.1: Ground Prism Layer Volume Mesh
Running Star-CCM+ for 1300 iterations, in order to have stationary and convergences solutions, the results are:

Drag value is of 6.58 N

Figure 5.2: GE Drag

This value is very close to B-2019 Drag Force value. With this results wheels influence on drag force is very small on the total value. In this way in the next section is analyzed a no-axissymetric body without wheels.

Figure 5.3: GE Wall Shear Stress

From WSS it could be possible analyze transiction point. It is located after the maximum diameter points, at almoust the 60% of the total lenght.

Speed and pressure values are in according to the phisycs problem. Maximum pressure at the leading edge and at the end, where there are two stopping points;
maximum speed and minimum pressure point is located in correspondence of B-2019 maximum diameter. Total pressure doesn’t change except for numerical errors that are brought to valley by the viscousity introducted by the numerical method.
5.0.2 Whithout Wheels: No Axissymmetric Body

The second way to analyze ground effect is possible thanks to the little effect of the wheels to the drag force value. In this study the body is no more an axissymmetric one but it is created with a loft operation between four splines. Each spline is bound to go through pedals, maximum, shoulders and inflection positions. Also they are tangent 90 grades to the nose.

X cordinates are set up on HEEDS best optimization design: C-Design. The optimization process will study the effect of top curvature due to the different bike heights at pedals, maximum diameter and shoulder position. Reports show that it is better to have a bicycle where the difference between heights is very small compared to the actual height. Difference between pedals and shoulders height must be smaller than 20 cm, with an 8° curvature. Top height and shoulders one must be similar (3~5 cm)
Wall Shear Stress show how laminar separation appear first on the body side and later on the top. This is caused by on top, the curvature is bigger than the side one. While the flow across the top has a favourite gradient pressure, on the body side the particelles already enters in the adverse gradient pressure zone.
This graphics show the previous discusses results. The difference between pedals and shoulders height must be smaller than 20 cm, with an 8° curvature. The difference between height and shoulders height must be very small (3~5 cm).
Chapter 6

Summary and Outlook

The present work illustrates shape optimization (using Star-CCM+ and HEEDS) by uncertainties in the nature of the transition from laminar to turbulent flow on axissymmetrical body as it invested by constant flow at zero grades incidence. A field function has been developed to take account of physics conditions associated to boundary layer transition. The body was located on air first and on ground later. Taking account of ground effect there are considered two difference approach: with or without wheels. The two methods lead to similar results. The calculation results still have to be validated experimentally. Especially it has to be investigated how surface irregularities of real cycle affect the transition location.
Bibliography


[8] CD-Adapco, STAR-CCM+ Guide

