Aerodynamic study of a photovoltaic solar tracker

Degree: Aerospace Engineering
Delivery date: 29-04-2016

Student:
Gutiérrez Castillo, Leonardo

Director:
Del Campo Sud, David
## Contents

### List of Tables

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>iii</td>
</tr>
</tbody>
</table>

### List of Figures

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>iv</td>
</tr>
</tbody>
</table>

### 1 Aim of the project

#### 4.1 Advantages of the study

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
</tr>
</tbody>
</table>

#### 4.2 Disadvantages of the study

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
</tr>
</tbody>
</table>

### 2 Scope of the project

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

### 3 Basic requirements of the project

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
</tr>
</tbody>
</table>

### 4 Justification of the study

#### 4.1 Advantages of the study

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
</tr>
</tbody>
</table>

#### 4.2 Disadvantages of the study

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
</tr>
</tbody>
</table>

### 5 State of the art

#### 5.1 Background

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
</tr>
</tbody>
</table>

##### 5.1.1 Photovoltaic trackers basic features

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
</tr>
</tbody>
</table>

##### 5.1.2 One and two axis trackers

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
</tr>
</tbody>
</table>

### 6 Previous Concepts

#### 6.1 Physics of the problem

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
</tr>
</tbody>
</table>

##### 6.1.1 Aerodynamic forces and moments

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
</tr>
</tbody>
</table>

##### 6.1.2 Basic equations

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
</tr>
</tbody>
</table>

#### 6.2 Definition of the problem

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
</tr>
</tbody>
</table>

##### 6.2.1 Products to analyse

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
</tr>
</tbody>
</table>

##### 6.2.2 Wind conditions or requirements

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
</tr>
</tbody>
</table>

##### 6.2.3 Cases of study

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>23</td>
</tr>
</tbody>
</table>

### 7 Analytic/experimental approach of the problem

#### 7.1 Simplification of the problem

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
</tr>
</tbody>
</table>

##### 7.1.1 Aerodynamic data

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
</tr>
</tbody>
</table>

##### 7.1.2 Wind profile

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>27</td>
</tr>
</tbody>
</table>

#### 7.2 Analytic results

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>27</td>
</tr>
</tbody>
</table>

### 8 CFD approach of the problem

#### 8.1 Fundamental concepts

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>29</td>
</tr>
</tbody>
</table>

##### 8.1.1 What is CFD?

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>29</td>
</tr>
</tbody>
</table>
List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1</td>
<td>One-axis trackers, basic construction features</td>
<td>9</td>
</tr>
<tr>
<td>6.1</td>
<td>Table of characteristics for the PVAST 327</td>
<td>20</td>
</tr>
<tr>
<td>6.2</td>
<td>Table of characteristics for the PVAST 196</td>
<td>21</td>
</tr>
<tr>
<td>6.3</td>
<td>Table of characteristics for the PVHT 2160</td>
<td>22</td>
</tr>
<tr>
<td>6.4</td>
<td>Table in which the velocities for each mode are specified</td>
<td>23</td>
</tr>
<tr>
<td>6.5</td>
<td>Table where the specific cases of study are exposed</td>
<td>24</td>
</tr>
<tr>
<td>7.1</td>
<td>Parameters for flat plate lift and drag from [19]</td>
<td>26</td>
</tr>
<tr>
<td>7.2</td>
<td>Lift and drag coefficients extracted from the aerodynamic data [17]</td>
<td>28</td>
</tr>
<tr>
<td>8.1</td>
<td>Turbulence models [21]</td>
<td>36</td>
</tr>
<tr>
<td>8.2</td>
<td>Comparison between $k - \omega$ and $k - \epsilon$ models</td>
<td>43</td>
</tr>
<tr>
<td>8.3</td>
<td>Recommended configurations for boundary conditions</td>
<td>61</td>
</tr>
<tr>
<td>9.1</td>
<td>Results of the PVAST 327, compared with the experimental data</td>
<td>74</td>
</tr>
<tr>
<td>9.2</td>
<td>Results of the PVAST 196, compared with the experimental data</td>
<td>74</td>
</tr>
<tr>
<td>9.3</td>
<td>Results of the PVHT 2160, compared with the experimental data</td>
<td>75</td>
</tr>
<tr>
<td>9.4</td>
<td>Total budget of the project</td>
<td>76</td>
</tr>
<tr>
<td>9.5</td>
<td>Brief tasks description</td>
<td>78</td>
</tr>
<tr>
<td>9.6</td>
<td>Interdependency relationship among tasks</td>
<td>79</td>
</tr>
<tr>
<td>9.7</td>
<td>Level of effort</td>
<td>80</td>
</tr>
</tbody>
</table>
List of Figures

5.1 Active tracker performance [12] ................................................................. 7
5.2 Passive tracker head in Spring/Summer tilt position [12] ........................... 8
5.3 Horizontal single axis tracker in Vellakoil, Tamil Nadu, India [12] .......... 9
5.4 Dual axis tracker mounted on a pole [12] ..................................................... 10
6.1 Pressure and shear stress [1] ...................................................................... 12
6.2 Resultant aerodynamic force and the components into which it splits [1] ... 12
6.3 The physical origin of moment on an aerodynamic shape [1] ................. 13
6.4 Some areas and lengths of reference ......................................................... 15
6.5 PVAST 327 back view [20] ..................................................................... 20
6.6 PVAST 196 back view [20] ............................................................... 21
6.7 PVHT 2160 back view [20] ............................................................... 22
7.1 (a) Free stream lift and (b) drag as a function of AR and α [17] .......... 27
8.1 The Reynolds experiment where laminar and turbulent flow are represented. 34
8.2 Creation of eddies behind a cylinder ......................................................... 35
8.3 Extend of modeling for certain types of turbulent models .................. 37
8.4 Plot of the Reynolds decomposition ....................................................... 39
8.5 Transition from a laminar to a turbulent boundary layer ..................... 47
8.6 Representation of the sub-layers in a turbulent boundary layer .......... 49
8.7 Boundary layer separation .............................................................. 51
8.8 Predicting separation in a diffuser-type geometry .................................. 52
8.9 Elements used in a mesh or grid ............................................................. 53
8.10 Structured and unstructured mesh discretization .................................. 54
8.11 Example of a hybrid mesh ............................................................. 55
8.12 Types of boundary conditions ............................................................... 58
8.13 Configuration of the solar modules for the PVAST 327 solar tracker .... 63
8.14 Configuration of the solar modules for the PVAST 196 solar tracker .... 63
8.16 Configuration of the solar modules for the PVHT 2160 solar tracker .... 63
8.17 Representation of the fluid domain dimensions [13] ......................... 64
8.18 Fluid domain around the PVAST 327 modules in ANSYS ICEM CFD .... 64
8.18 Process of the blocking of the geometry ............................................. 65
8.19 Final blocking of the geometry ............................................................ 66
8.20 Exponential bunching at the boundary layer zone ............................ 67
8.21 Final mesh displayed in FLUENT ........................................................ 67
8.22 Final mesh with boundary conditions defined (in ICEM CFD) ............ 68
8.23 Settings in the General panel of FLUENT ............................................ 69
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.24</td>
<td>Selection of the turbulent model.</td>
<td>69</td>
</tr>
<tr>
<td>8.25</td>
<td>Inlet panel in FLUENT</td>
<td>70</td>
</tr>
<tr>
<td>8.26</td>
<td>Outlet panel in FLUENT</td>
<td>70</td>
</tr>
<tr>
<td>8.27</td>
<td>The Solution Methods panel in FLUENT.</td>
<td>71</td>
</tr>
<tr>
<td>8.28</td>
<td>Last step before the calculations.</td>
<td>72</td>
</tr>
<tr>
<td>9.1</td>
<td>Pressure distribution in surface (0°)</td>
<td>73</td>
</tr>
<tr>
<td>9.2</td>
<td>Pressure distribution in surface (30°)</td>
<td>73</td>
</tr>
<tr>
<td>9.3</td>
<td>Pressure contour in the mid-plane (60°)</td>
<td>73</td>
</tr>
<tr>
<td>9.4</td>
<td>Pressure contour in the mid-plane (90°)</td>
<td>73</td>
</tr>
<tr>
<td>9.5</td>
<td>Pressure contour in the mid-plane (30°)</td>
<td>74</td>
</tr>
<tr>
<td>9.6</td>
<td>Pressure contour in the mid-plane (60°)</td>
<td>74</td>
</tr>
<tr>
<td>9.7</td>
<td>Turbulence viscosity in a intersecting mid-plane (20°)</td>
<td>75</td>
</tr>
<tr>
<td>9.8</td>
<td>Eddy viscosity in the mid-surface (60°)</td>
<td>75</td>
</tr>
<tr>
<td>9.9</td>
<td>Eddy viscosity contour at mid-plane (0°)</td>
<td>76</td>
</tr>
<tr>
<td>9.10</td>
<td>WBS diagram for the aerodynamic study</td>
<td>78</td>
</tr>
</tbody>
</table>
Section 1: Aim of the project

1 Aim of the project

The main aim of this project will be the determination of the aerodynamic features of ground-mounted solar trackers under atmospheric boundary layer flows and the study and identification of the aerodynamic interactions of the solar trackers when they are displayed as an array.
2 Scope of the project

Here, a list of the tasks that will be achieved throughout the development of the project will be presented, some of them will be eventually turn out to be deliverable documents or sections of the final project document.

- **Background:** A concise description of the need that is being covered and the advantages or disadvantages of the approach used.
- **State of the art:** A description of the current level of development in the field of solar trackers design.
- **Previous concepts:** Some basic concepts will be introduced in order to understand the physics of the problem.
- **Analytic approach:** From an analytical point of view, an approach of the problem will be made, this will be later compared with the numerical solution.
- **CFD approach:** It will be intended to explain some of the fundamental concepts which has to be taken into account when it comes to the numerical simulation of a fluid mechanics problem.
- **Aerodynamic study:** Once all the concepts behind a CFD study are explained, the study itself, which includes the creation of the geometry, the meshing, the boundary conditions, etc..., can be performed.
- **Results and comparison:** Validation and discussion of the results.
- **Conclusions:** Once the results are interpreted from different points of view some conclusions can be made and also recommendations to continue with the study can be done.
3 Basic requirements of the project

The aerodynamic study of the solar tracker must fulfil the following requirements:

- The conditions at which the model is studied must be the most realistic as possible, taking into account for which terrain is it thought to be installed.
- The results must be validated and if possible compared.
- The results of the study must be reliable, meaning that all assumptions made in the approach have to be justified.
- It will be considered that the aerodynamic performance of the photovoltaic solar tracker is acceptable, if the associated loads are within the specifications of the products offered by the solar trackers global market.
- Improvements proposal must be done with the economical aspect as a fundamental decision criteria.
4 Justification of the study

In order to justify the usefulness of the project it will be presented the advantages of the study but to not be impartial the other side will be presented also, this is the disadvantages or drawbacks that the study has.

4.1 Advantages of the study

The aerodynamic study will be done using CFD (*Computational Fluid Dynamics*) methods and FEA (*Finite Element Analysis*) software, this means that no prototype will have to be constructed and therefore no money has to be spent, in order to produce this prototype. This would have to be done in the case that the project was based on an experimental study. Since the company is a small one, it is better to perform this type of study before undertaking the manufacturing of a prototype.

Regarding the use of CFD in this study, these advantages are inherent to this kind of method [21]:

- Substantial reduction of lead times and costs of new designs.
- Ability to study systems where controlled experiments are difficult or impossible to perform (e.g. very large systems).
- Ability to study systems under hazardous conditions at and beyond their normal performance limits (e.g. safety studies and accident scenarios).
- Practically unlimited level of detail of results.
- The variable cost of an experiment, in terms of facility hire and/or man-hour costs, is proportional to the number of data points and the number of configurations tested. In contrast, CFD codes can produce extremely large volumes of results at virtually no added expense and it is very cheap to perform parametric studies, for instance to optimise equipment performance.

CFD is of great utility and a great design tool. However, it is a tool that can not be used to design the final product but to design the preliminary or first design since in every design project the final product will originate from the tests and improvements of earlier designs starting from the preliminary design or prototype.
4.2 Disadvantages of the study

As it has been stated before, one of the drawbacks when speaking of using *Computational Fluid Dynamics* methods is that depending on the complexity of the model some considerable computing power is required and therefore the computing resources that are available are of great relevance in this kind of analysis.

More generally, when speaking of an organisation which uses this kind of method (CFD) it implies a substantial investment outlay due to the need of qualified people to run the software and communicate their results, and a consideration of the modelling skills required by CFD users.

Furthermore, the results of the study can not be taken instantly as true, as with all FEA (*Finite Element Analysis*) studies, the results will depend on the mesh and some other variables and conditions that may vary the final results very drastically, in order to validate the results, it will be compared with some references being they either analytical or experimental.
5 State of the art

The project which is going to be developed will be specifically speaking a study in which a analysis, a discussion, an evaluation of alternatives and a assessment of the results will be made regarding factors of design, planning, production methods, management and use of resources, etc., related all of them with the knowledge fields of the degree.

More specifically the study will consist in the application of Computer Fluid Dynamics (CFD) techniques with the objective to make an estimation of the aerodynamic loads to which is being submitted a solar tracker in an open field, which is the common installation location for this kind of devices. Through the development of this study the conditions of the flow at which the supporting structure of the PV solar tracker is placed, will be known, this is a valuable information since it allows to make improvements to the design once knowing its aerodynamics limitations.

In order to know in a better way the object of study, which in this study will be a solar tracker, it will be presented in a brief review the photovoltaic systems used nowadays.

5.1 Background

A solar tracker is a device that orients a payload into the sun. This payload can be solar panels, parabolic troughs, fresnel reflectors, mirrors or lenses. However for this study, solar panels will be considered as payload since this is the configuration that the company sells as a product, so its study is of great interest for them.

For flat-panel photovoltaic systems, trackers are used to minimize the angle of incidence between the incoming sunlight and a photovoltaic panel. This increases the amount of energy produced from a fixed amount of installed power generating capacity, since the tracking of the sun improves the efficiency of whole photovoltaic system due to the increase in solar radiation that the solar panel is able to absorb.

5.1.1 Photovoltaic trackers basic features

Considering basic construction principles trackers can be divided into active and passive trackers [12]:

Active trackers

Active trackers use motors and gear trains to direct the tracker as commanded by
a controller responding to the solar direction. This response is created by using a light-sensing device, typical trackers have two or more photosensors such as photodiodes, configured differentially so that they output a null when receiving the same light flux (fig. 5.1). In order to control and manage the movement of these massive structures special slewing drives are designed and rigorously tested. The technologies used to direct the tracker are constantly evolving and recent developments have included the use of wire-ropes and winches to replace some of the more costly and more fragile components.

![Figure 5.1: Active tracker performance](image)

The ideal would be that the motors consume the less energy possible, that is using them only as necessary. So in most cases, instead of a continuous motion the tracker is moved in discrete steps. It must be taken into consideration also that if the light is below some threshold there would not be enough power generated to warrant reorientation, so some consideration must be made to keep the tracker from wasting energy during cloudy periods.

**Passive trackers**

Passive solar trackers are based on thermal expansion of matter (freon, a low boiling point compressed gas fluid) or on shape memory alloys. They are usually composed of a couple of actuators working against each other which are balanced by equal illumination. By differential illumination of actuators, the resulting unbalance of forces (gas pressure or dilatation forces created by solar heat) is used for orientation of the apparatus in such a direction where equal illumination of actuators and balance of forces is restored.
Passive solar trackers, compared to active trackers, are less complex but they work with lower efficiency and at low temperatures do not work at all, also, as this is a non-precision orientation it is unsuitable for certain types of concentrating photovoltaic collectors but works fine for common PV panel types.

Chronological trackers

A chronological tracker counteracts the Earth’s rotation by turning at the same speed as the Earth relative to the Sun, around an axis parallel to the Earth’s, but in the direction opposite to the Earth’s rotation. To do this, a simple rotation mechanism, turning at a constant speed of one revolution per day or 15 degrees per hour, is adequate for many purposes, such as keeping a photovoltaic panel pointing within a few degrees of the Sun, but for accurate tracking, such as may be needed to keep a telescope aimed at the Sun, the equation of time must be taken into account, so the tracker moves according to apparent solar time, often called 'sundial time'.

A chronological tracker is a very simple yet potentially a very accurate solar tracker specifically for use with a polar mount. In theory the tracker may rotate completely, assuming there is enough clearance for a complete rotation, and assuming that twisting wires are not an issue, otherwise a simple reset to the dawn position may be performed at anytime between dusk and dawn.

Manual trackers

In some developing nations, drives have been replaced by operators who adjust the trackers. This has the benefits of robustness, having staff available for maintenance and creating employment for the population in the vicinity of the site.
5.1 Background

5.1.2 One and two axis trackers

All the trackers presented earlier can be classified regarding its degrees of freedom in two types:

**One axis trackers**

Different one-axis trackers solutions are available on the market. In table below different basic features of different one-axis tracker designs are presented.

<table>
<thead>
<tr>
<th>Tilted N-S axis tracker</th>
<th>Rotation axis is tilted.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polar axis tracker</td>
<td>Axis tilt equals local latitude, incidence angle equals declination</td>
</tr>
<tr>
<td>Horizontal N-S axis tracker</td>
<td>No shadows in the N-S direction, low wind profile, suitable for flat land</td>
</tr>
<tr>
<td>Azimuth axis tracker</td>
<td>Adapt well to uneven land</td>
</tr>
</tbody>
</table>

Table 5.1: One-axis trackers, basic construction features [7]

![Horizontal single axis tracker in Vellakoil, Tamil Nadu, India](image)

Figure 5.3: Horizontal single axis tracker in Vellakoil, Tamil Nadu, India [12].

**Two axis trackers**

Two-axis tracker products include azimuth-elevation trackers and declination-hour angle trackers. Azimuth trackers can be pedestal mounted or with multiple support or wheel type support. In case of multiple support points wind resistance is better than for pedestal type trackers.
Once the diverse types of solar tracker are explained the specific object of study is defined here. The products that the company offers in terms of solar trackers are mainly one model of a horizontal North-South axis tracker and three different models of two axis pedestal mounted trackers which differs between them in size and number of solar panels.
6 Previous Concepts

Before entering in the explanation of specific concepts related with the *Computational Fluid Dynamics* analysis, a brief exposition of the basic concepts in which is based the study has to be done, as an introductory step, this will be helpful since the framework is settled and from now on aspects with more complexity can be exposed without assuming previous knowledge is already known.

6.1 Physics of the problem

In order to explain the basic physics of this problem, first it will be exposed the basic principle on which the phenomenon that is going to be studied is based, for doing so, some basic fluid mechanics concepts and also some simplifications of the problem will be used in order to make a more clear explanation. The problem as it is, will be defined also in this section, setting the dimensions of the object of study and clarifying which will be the wind and atmospheric conditions under which the solar tracker will be brought.

6.1.1 Aerodynamic forces and moments

The aerodynamic forces and moments on a body are due to only two basic sources:

1. *Pressure distribution* over the body surface.

2. *Shear stress* distribution over the body surface

No matter how complex the body shape may be, the aerodynamic forces and moments on the body are due entirely to the above two basic sources. The only mechanisms nature has for communicating a force to a body moving through a fluid are pressure and shear stress distributions on the body surface. Both pressure \( p \) and shear stress \( \tau \) have dimensions of force per unit area (pounds per square foot or newtons per square meter). As sketched in figure 6.1, \( p \) acts normal to the surface, and \( \tau \) acts tangential to the surface. Though it has not been explicit mentioned earlier, shear stress is of great relevance when it comes to the estimation of the aerodynamic forces, and it is caused by friction between the body and the air.
Since the fluid is in motion, we can define a flow direction along the motion. The component of the net force perpendicular (or normal) to the flow direction is called the lift; the component of the net force along the flow direction is called the drag. These are definitions. In reality, there is a single, net, integrated force caused by the pressure variations along a body (fig. 6.2). This aerodynamic force acts through the average location of the pressure variation which is called the center of pressure. Note that in fig. 6.2 the resultant aerodynamic force is also split in an alternative way, that is the components parallel and perpendicular to the chord line, in the case of an airfoil, or a reference axis of an aerodynamic shape generally speaking. These components are called the normal force and axial force and are denoted by $N$ and $A$, respectively. It is useful to be familiar with both systems tough normal and axial forces are nowadays a little deprecated since $L$ and $D$ are almost always the system used by choice.

In addition to lift and drag, the surface pressure and shear stress distributions create a moment
6.1 Physics of the problem

M that tends to rotate the wing, speaking of an airplane, or in the case of a solar tracker structure, the photovoltaic panels. To see more clearly how this moment is created, it has to be taken into consideration the surface pressure distribution over an airfoil, in which this difference of pressure can be seen more clearly.

As sketched in fig. 6.3 (ignoring the shear stress for this discussion), considering just the pressure on the top of the surface of the airfoil, this pressure gives rise to a net force $F_1$ in the general downward direction. Moreover, $F_1$ acts through a given point on the chord line, point 1, which can be found by integrating the pressure times distance over the surface (analogous to finding the centroid or center of pressure from integral calculus). On the other side, considering just the pressure on the bottom surface of the airfoil, this gives rise to a net force $F_2$ in the general upward direction, acting through point 2.

![Figure 6.3: The physical origin of moment on an aerodynamic shape](image)

The total aerodynamic force on the airfoil is the summation of $F_1$ and $F_2$, and lift is obtained when $F_2 > F_1$. However, $F_1$ and $F_2$ will create a moment that will tend to rotate the airfoil. The value of this aerodynamically induced moment depends on the point about which it is chosen to take moments. Intuition will tell that lift, drag and moments will change as the angle of attack $\alpha$ of the body changes. However, there exists a certain point on the body about which moments essentially do not vary with $\alpha$. This point is defined as the aerodynamic center, and the moment about the aerodynamic center is designated $M_{ac}$. By definition,

$$M_{ac} = \text{const}$$  \hspace{1cm} (6.1)

The location of the aerodynamic center for real aerodynamic shapes can be found from experiment, and in this case it will be of relevance since it is a point which eases the analysis of forces at different angles of attack. This is due to the fact that when the angle of attack changes, the pressure distribution changes (implying a displacement of the center of pressure) and thus
the force and torque generated also changes, so determining the aerodynamic behaviour of a aerodynamic shape is very complicated if the center of pressure is used to analyse the forces.

Using the aerodynamic center as the location where the aerodynamic force is applied eliminates the problem of the movement of the center of pressure with angle of attack in aerodynamic analysis.

**Lift, drag, and moment coefficients**

As the discussion of aerodynamics progresses, it will become clear that there are quantities of an even more fundamental nature than the aerodynamic forces and moments themselves. These are the *dimensionless force and moment coefficients* found through the use of *dimensional analysis*, which in engineering and science is a tool that allows to simplify the study of any phenomenon in which many physical magnitudes are involved as independent variables. Its fundamental result, the *Buckingham π theorem* can change the original set of input dimensional parameters of a physics problem by another set of input dimensionless parameters more reduced.

Let $\rho_\infty$ and $V_\infty$ be the density and velocity, respectively, in the free-stream, far ahead from the body. A dimensional quantity called the free-stream *dynamic pressure* is defined as:

$$q_\infty = \frac{1}{2} \rho_\infty v_\infty^2 \quad (6.2)$$

The dynamic pressure has the units of pressure (i.e. pounds per square foot or newtons per square meter. In addition, let $S$ be a reference area and $l$ be a reference length. The dimensionless force and moment coefficients are defined as follows:

- **Lift coefficient**:
  $$C_L = \frac{L}{q_\infty S}$$

- **Drag coefficient**:
  $$C_D = \frac{D}{q_\infty S}$$

- **Normal coefficient**:
  $$C_N = \frac{N}{q_\infty S}$$

- **Axial coefficient**:
  $$C_A = \frac{A}{q_\infty S}$$
6.1 Physics of the problem

Moment coefficient:

\[ C_M = \frac{M}{q_\infty \ S \ l} \]

In the above coefficients, the reference area \( S \) and reference length \( l \) are chosen to pertain to the given geometric shape, for different shapes, \( S \) and \( l \) may be different things. For example, for an airplane wing, \( S \) is the planform area, and \( l \) is the mean chord length (fig.6.4(a)); for a sphere, \( S \) is the cross sectional area, and \( l \) is the diameter (fig.6.4(b)). When dealing with a solar tracker body, which is this case, \( S \) could be the total area of the solar panels, and \( l \) the length of them, although when it comes the moment to define it, it will be explicitly commented.

![Figure 6.4: Some areas and lengths of reference.](image)

6.1.2 Basic equations

The basic equations governing the fluid flow will be presented as well as the basic conservation principles and laws used to obtain them.

Starting with the basics, a fluid is a substance which deforms continuously under an applied shear stress, no matter how small this is. In this study a Newtonian incompressible fluid will be used. In the case of Newtonian fluids there is a lineal relationship between the stress and rate of strain so that:

\[ \tau = \mu \frac{du}{dy} \]  

\[ (6.3) \]
Where:

\( \tau \) : Shear stress.

\( \mu \) : Viscosity.

\( \frac{du}{dy} \) : Velocity gradient.

One of the most important properties of a fluid is the viscosity. The viscosity is a property of the fluid, which measures the resistance of the fluid of being deformed by the shear stress. The fluid adheres to the wall, so that its velocity in the wall’s surface is zero. In many flows, the effects of viscosity are important only near walls, so that the flow in the largest part of the domain is considered as inviscid.

Due to the three fundamental physic principles which all fluid dynamics satisfy,

1. The mass of a fluid is conserved.
2. The momentum is conserved (Newton’s second law).
3. The Energy is conserved

the governing equations of fluid dynamics can be obtained \[21\].

**Conservation of mass**

\[
\frac{Dm}{Dt} = 0 \quad (6.4)
\]

The mass balance for a fluid element can be written down as:

<table>
<thead>
<tr>
<th>Rate of increase</th>
<th>Net rate of flow of</th>
</tr>
</thead>
<tbody>
<tr>
<td>of mass in fluid</td>
<td>mass into fluid element</td>
</tr>
</tbody>
</table>

After doing some mathematical calculations we get to the **continuity equation**:

\[
\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{u} = 0 \quad (6.5)
\]

For an incompressible fluid, where \(-\) is a constant, the equation becomes
6.1 Physics of the problem

\[ \nabla \cdot \bar{u} = 0 \quad (6.6) \]

Conservation of momentum

Newton’s second law states that the rate of change of momentum of a fluid particle is equal to the sum of the forces on the particles.

<table>
<thead>
<tr>
<th>Rate of increase of momentum of fluid particle</th>
<th>Sum of forces on fluid particle</th>
</tr>
</thead>
</table>

The momentum equations can be written as:

\[ \frac{\partial (\rho \bar{u})}{\partial t} + \rho (\bar{u} \cdot \nabla) \cdot \bar{u} = -\nabla p + \nabla \cdot \tau + F \quad (6.7) \]

These are the well-known Navier-Stokes equations which describe how the velocity, pressure, temperature and density of a moving fluid are related. One the left-hand side of the equation are the derivative over time and the internal forces (convection term). In the right-hand side we can see the gradient of the pressure, the viscous forces and the gravitational forces.

Very often the term of Navier-Stokes equations is applied as well to the group of equations containing the Navier Stokes equations themselves and the continuity equation and the energy equation (presented in the following).

Conservation of energy

The first law of thermodynamics states that the rate of change of energy of fluid particle is equal to the rate of heat addition to the fluid particles plus the rate of work done on the particle.

<table>
<thead>
<tr>
<th>Rate of increase of energy of fluid particle</th>
<th>Net rate of heat added to fluid particle</th>
<th>Net rate of work done on fluid particle</th>
</tr>
</thead>
</table>
\( \rho \frac{DE}{Dt} = -\nabla \cdot (p\vec{u}) + \left[ \frac{\partial (u\tau_{xx})}{\partial x} + \frac{\partial (u\tau_{yx})}{\partial y} + \frac{\partial (u\tau_{zx})}{\partial z} + \frac{\partial (v\tau_{xy})}{\partial x} + \frac{\partial (v\tau_{yz})}{\partial y} + \frac{\partial (w\tau_{xz})}{\partial z} \right] + \nabla \cdot (k\nabla \cdot T) + S_E \) (6.13)

Where \( E = \frac{e^2}{2} \), with \( e \) the sum of internal energy, \( k \) is the thermal conductivity and \( S_E \) represents a source of energy.

Taking a look to all of the equations written above some comments and observations can be made.

The governing equations are a system of nonlinear partial differential equations, and they are very difficult to solve analytically. As is well known, there are few exact solutions, and all of these have been obtained introducing simplifying assumptions. To date, there is no general closed-form solution to these equations.

**Summary of equations in fluid mechanics**

**Continuity equation**

\[ \frac{D\rho}{Dt} + \rho \nabla \cdot \vec{u} = 0 \] (6.9)

**Navier-Stokes equations:**

\[ \rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + F_x \] (6.10)

\[ \rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + F_y \] (6.11)

\[ \rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + F_z \] (6.12)

**Energy equation:**

\[ \rho \frac{DE}{Dt} = -\nabla \cdot (p\vec{u}) + \left[ \frac{\partial (u\tau_{xx})}{\partial x} + \frac{\partial (u\tau_{yx})}{\partial y} + \frac{\partial (u\tau_{zx})}{\partial z} + \frac{\partial (v\tau_{xy})}{\partial x} + \frac{\partial (v\tau_{yz})}{\partial y} + \frac{\partial (w\tau_{xz})}{\partial z} \right] + \nabla \cdot (k\nabla \cdot T) + S_E \] (6.13)
6.2 Definition of the problem

This is a system of 5 equations with 7 unknowns ($\rho$, $p$, $T$, $e$, $u$, $v$, $w$). To close the entire system of equations two more equations must be added. The equations we must add are the equation of state which relates the pressure, temperature and density of the fluid and a thermodynamic relation between state variables. These equations depend on each case.

6.2 Definition of the problem

Once the basics of the physics involved in the situation of study are in a concise manner explained, the next step is getting down to define specifically how the objects of study will be, this includes its dimensions, the main differences between them, its specifical characteristics and what is it wanted to achieve when it comes to the production of the different products.

6.2.1 Products to analyse

In general and as the title for this project says what is going to be studied will be a variety of solar trackers, two dual-axis solar trackers and one single-axis tracker. Coming next, the designation and the characteristics for every product will be defined.

**PVAST 327**

*PVAST 327* is a fully integrated 32.7 $m^2$ aperture photovoltaic tracker with a total of 5.3 kWp of power. Specifically speaking it is a pole mounted two-axis tracker using for this purpose high accuracy actuators and stepper motors. The tracking of the solar radiation in this case is done by means of a simple rotation mechanism, which moves the tracker approximately at 15 degrees per hour according to the 'sundial time'. This tracking manoeuvre is done in a continuous manner achieving with this a increased reliability and a reduced consumption [20].

Its installation can be done by driving the pole into the ground which is a very cheap process for large installations, as it is done using automated equipment. Also it implies low environmental impact as it can be dismantled using the same machines. For small installations as well as for the top of flat rooftops concrete foundations can also be used. It can be assembled at decentralised facilities and can be transported to the field as a single unit, thus optimizing logistics and minimizing installation and commissioning costs.
Aerodynamic Study of a PV Solar Tracker – Report

Figure 6.5: PVAST 327 back view [20].

<table>
<thead>
<tr>
<th>Solar aperture</th>
<th>$32.7 \ m^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supermodule dimensions</td>
<td>4965 mm height, 3330 mm width</td>
</tr>
<tr>
<td>Supermodule Aspect ratio ($AR$)</td>
<td>0.67</td>
</tr>
<tr>
<td>PV cells</td>
<td>156 mm square, poly-crystalline</td>
</tr>
<tr>
<td>Cells per module</td>
<td>60</td>
</tr>
<tr>
<td>Modules per system</td>
<td>20</td>
</tr>
<tr>
<td>Elevation span</td>
<td>0/90 degrees</td>
</tr>
<tr>
<td>Azimuth span</td>
<td>0/360 degrees</td>
</tr>
</tbody>
</table>

Table 6.1: Table of characteristics for the PVAST 327 [20].

The simulations will be done for the supermodules (which in the case of the PVAST 327 consists of 10 modules), this is why in the table 6.1 the dimensions and its aspect ratio ($AR = \frac{\text{width}}{\text{height}}$) are specified.

The drawings for this solar tracker are included in the drawings annex.

PVAST 196

PVAST 196 is a fully integrated 19,6 $m^2$ aperture photovoltaic tracker with a total of 3,18 kWp of power. In a simpler way it is a small version of the PVAST 327, including this 20 solar modules, 10 at each side of the horizontal tube arranged in 5 lines of 2 modules each one, whereas the PVAST 196 has only 12 solar modules, here lies the fact of the reduction of power.
6.2 Definition of the problem

It also has all the features earlier stated for the PVAST 327, its installation characteristics are the same and uses the same technology for the tracking manoeuvre.

![Figure 6.6: PVAST 196 back view][20].

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solar aperture</td>
<td>19.6 $m^2$</td>
</tr>
<tr>
<td>Supermodule dimensions</td>
<td>2978 mm height, 3330 mm width</td>
</tr>
<tr>
<td>Supermodule Aspect ratio (AR)</td>
<td>1.12</td>
</tr>
<tr>
<td>PV cells</td>
<td>156 mm square, poly-crystalline</td>
</tr>
<tr>
<td>Cells per module</td>
<td>60</td>
</tr>
<tr>
<td>Modules per system</td>
<td>12</td>
</tr>
<tr>
<td>Elevation span</td>
<td>0/90 degrees</td>
</tr>
<tr>
<td>Azimuth span</td>
<td>0/360 degrees</td>
</tr>
</tbody>
</table>

Table 6.2: Table of characteristics for the PVAST 196[20].

The drawings for this solar tracker are included in the drawings annex.

PVHT 2160

PVHT 2106 is a fully integrated 210.6 $m^2$ aperture photovoltaic single axis horizontal tracker with a total of 34 kWp. As well as the other products it uses a continuous astronomical tracking control system for increased reliability and reduced consumption. In its standard configuration, the one that is being offered by the company, it comprises 12 segments of 9 modules per segment, each segment based on a structural square steel tube. Its main assembly steps consist in riveting
components together (wings to central square tubes and modules to wings), speeding up system installation.

![PVHT 2160 back view](image)

**Figure 6.7: PVHT 2160 back view [20].**

<table>
<thead>
<tr>
<th>Solar aperture</th>
<th>210.6 $m^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supermodule dimensions</td>
<td>2978 $mm$ height, 5904 $mm$ width</td>
</tr>
<tr>
<td>Supermodule Aspect ratio ($AR$)</td>
<td>2.01</td>
</tr>
<tr>
<td>PV cells</td>
<td>156 $mm$ square, poly-crystalline</td>
</tr>
<tr>
<td>Cells per module</td>
<td>72</td>
</tr>
<tr>
<td>Modules per system</td>
<td>108</td>
</tr>
<tr>
<td>Tilting span</td>
<td>-60/+60 degrees</td>
</tr>
</tbody>
</table>

Table 6.3: Table of characteristics for the *PVHT 2160* [20].

The drawings for this solar tracker are included in the drawings annex.

### 6.2.2 Wind conditions or requirements

When speaking of the range of wind velocities within which the solar trackers have to withstand the aerodynamic forces without collapsing or suffering a plastic deformation, it is appropriate that a distinction between the modes in which the machines will be operating can be considered prior to state a set of values or a range of acceptable wind velocities.

There are two main modes of operation for the solar trackers, the *Tracking Mode* and the *Protection Mode*, the first of them is the mode in which anyone would expect a solar tracker to be working, as it name indicates in this working mode the solar tracker is executing the tracking of the solar radiation very slowly as the sun moves following its natural cycle. The *Protection Mode* is a special mode where due to continuos and strong wind gusts the *Tracking*
6.2 Definition of the problem

*Mode* is disabled and the solar tracker change its position to a one where the solar panels are oriented parallel to the ground in order to not block the wind flow, that is generating the smaller drag force possible, nonetheless depending of the incidence of the wind an aerodynamic force could be still generated, being this an important aspect to take into account when doing the simulations.

The next table presents the range of velocities which the two working modes comprise and the limit velocity for every mode.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Range of velocities</th>
<th>Limit velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tracking mode</td>
<td>0 - 80 km/h</td>
<td>140 km/h</td>
</tr>
<tr>
<td>Protection mode</td>
<td>80 - 140 km/h</td>
<td>180 km/h</td>
</tr>
</tbody>
</table>

Table 6.4: Table in which the velocities for each mode are specified.

These values are stated by the company and are based in the actual state of the art, that is the characteristics of the different products which companies within the solar energy business are offering nowadays, so the simulations that will be carried out will take into consideration these velocities since the point of interest of this study lies in the fact that the solar tracker can withstand critic velocities. Having in mind what has been just said, the velocities of interest which will be useful when stating the conditions for the aerodynamic simulations are the limit velocities, being these the maximum velocities at which the structure presents deformations that are within the elastic regime, no plastic deformations are allowed no matter how minimal they are.

6.2.3 Cases of study

Stating the basic conditions at which the simulations will be carried out is a good way to clarify some parameters from the set-up of the study, this is very helpful since there are a lot of factors and parameters that have to be defined before the running of the simulation.

As it have been commented before, the models will be proved in two configurations, the working and the protection configuration. For each configuration several analysis will be done some of them at different angles of incidence just to do a comparison and to find a critical work point in which high wind gusts implies a terrible effect on the structure of the solar tracker.

The next table represents in a explicit manner, which will be the analysis that will be due to term a priori.
Table 6.5: Table where the specific cases of study are exposed.

<table>
<thead>
<tr>
<th>Model</th>
<th>Configuration</th>
<th>Speed Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>PVAST 327</td>
<td>Working config.</td>
<td>140 km/h (limit velocity).</td>
</tr>
<tr>
<td></td>
<td>Protection config.</td>
<td>180 km/h (limit velocity).</td>
</tr>
<tr>
<td>PVAST 196</td>
<td>Working config.</td>
<td>140 km/h (limit velocity).</td>
</tr>
<tr>
<td></td>
<td>Protection config.</td>
<td>180 km/h (limit velocity).</td>
</tr>
<tr>
<td>PVHT 2160</td>
<td>Working config.</td>
<td>140 km/h (limit velocity).</td>
</tr>
<tr>
<td></td>
<td>Protection config.</td>
<td>180 km/h (limit velocity).</td>
</tr>
</tbody>
</table>
7 Analytic/experimental approach of the problem

In order to estimate in a very simple but consistent manner the aerodynamic forces generated in an open field, a study of the problem with an analytic perspective will be done. For doing so, some simplifications have to be taken into consideration, this is done with the intention of reducing the degree of complexity presented a priori by this case. One of the main difficulties when modelling a case of these characteristics for a CFD analysis is the consideration of the turbulence and the simulation of it, as it will be seen in following sections, the selection of the turbulence model is not a trivial decision. This and other complex concepts will be simplified in this section.

7.1 Simplification of the problem

When it comes to define how the object of study, in this case the solar panels, is going to be assimilated in order to obtain a experimental or empirical data, a brief review of the common aerodynamic shapes is enough to come up with the solution.

7.1.1 Aerodynamic data

There is a vast spectrum from which some aerodynamic data can be used. Most wind engineering studies of photovoltaic modules ([18] and [16]) have measured the average pressure distribution on model modules and arrays and inferring the forces. Some authors, for example, [4], measured the moments as well, but the data is limited: the pitch (overturning) moment has not been reported.

In many cases, it has be found that photovoltaic modules can be seen as flat plates, at least aerodynamically speaking. This is an assumption that eases the research of aerodynamic data, being it either analytic equations or experimental results.

In particular, [19], directly measured the forces on low aspect ratio (AR) rectangular flat plates (among other geometries) for $0.5 \leq AR \leq 2$ and $\alpha \leq 50^\circ$. Their Equations are data correlations for the lift and drag respectively. These equations are:

\[ C_L = \sin \alpha \cos \alpha (K_p \cos \alpha + \pi \sin \alpha) \] (7.1)

and:

\[ C_D = C_{D0} + KC_L^2 \] (7.2)
Where $C_{D0}$ is the zero-angle drag due to viscosity, and the constants $K_p$ and $K$ are given in Table 7.1 along with $\alpha_m$, the highest angle for which the equations are valid. At low $\alpha$, the viscosity dominates the drag, with typical values of $C_{D0}$ being $0.015$ (Torres and Mueller [REF]). As $\alpha$ increases, however, it is to be expected that pressure becomes dominant, and that eventually:

$$C_D/C_L = \tan(\alpha) \quad (7.3)$$

and equations 7.1 and 7.2 will no longer be valid. In other words, $\alpha_m$, is a lower bound on the angles for which Equation 7.3 should be valid and all forces on the plate are due to pressure acting in the direction normal to the plate surface. Table 7.1 shows that $\alpha_m$ is strongly dependent on $AR$, and implies that Equation 7.3 holds over an increasing range of $\alpha$ as $AR$ increases.

<table>
<thead>
<tr>
<th>AR</th>
<th>$\alpha_m$</th>
<th>$K_p$</th>
<th>$K$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>35</td>
<td>0.831</td>
<td>0.67</td>
</tr>
<tr>
<td>0.75</td>
<td>33</td>
<td>1.26</td>
<td>0.565</td>
</tr>
<tr>
<td>1.0</td>
<td>28</td>
<td>1.59</td>
<td>0.53</td>
</tr>
<tr>
<td>1.25</td>
<td>20</td>
<td>1.85</td>
<td>0.483</td>
</tr>
<tr>
<td>1.5</td>
<td>15</td>
<td>2.10</td>
<td>0.417</td>
</tr>
<tr>
<td>1.75</td>
<td>14</td>
<td>2.35</td>
<td>0.409</td>
</tr>
<tr>
<td>2.0</td>
<td>13</td>
<td>2.59</td>
<td>0.374</td>
</tr>
</tbody>
</table>

Table 7.1: Parameters for flat plate lift and drag from [19].

This is a good analytical estimation for the $C_L$ and the $C_D$, from which the aerodynamic forces can be deduced. However, for high $\alpha$ the analytical results obtained from these equations are out of the application range and therefore they are totally wrong. Given that results at high $\alpha$ are needed, since the main aim for these calculations is having some reference in order to do a comparison between numerical and analytic results, the best option is to deduce the loads for the supermodule’s solar panels by means of the following experimental data.
7.2 Analytic results

![Figure 7.1: (a) Free stream lift and (b) drag as a function of AR and α][17.]

### 7.1.2 Wind profile

The problem as it presents now is the one of a flat plate in a fluid flow with a certain mean velocity, this fluid flow has to be as well simplified. Having in mind the existence of the atmospheric boundary layer (from the earth’s surface to around 2000 meters) one could think that it is mandatory to contemplate the inclusion of this fact in this study.

However, in the experiment from where the data is extracted no attempt was made to accurately simulate the atmospheric boundary layer when obtaining the experimental data because this would have required the use of very small flat plates that had low Reynolds numbers and were difficult to measure accurately.

### 7.2 Analytic results

In the table 7.2 the analytical coefficients and center of pressures are shown.
The experimental data that is used here was obtained for Reynolds numbers (a dimensionless parameter that characterizes the behaviour of the flow, it is presented in section 8.1.3) slightly lower than those at which the simulations will be carried out. However, considering that the difference is, at most, of only one order of magnitude and that with all the assumptions already made there is no sense in being so scrupulous and finding experimental data that is obtained at the same Reynolds number. Furthermore, it is clear that these analytical results will be only used as a validation for the numerical results.
Section 8: CFD approach of the problem

8 CFD approach of the problem

The main aim of this study is, in the end, to obtain valid numerical results so that the phenomenon is simulated in the best and most practical way possible, this means that perfection and high accuracy are not sought in the simulations that will be performed, far from this, the best result would be that in which the final conclusions are not so far from the reality and achieved without reconsidering drastic changes in the set-up and the conditions of the study. There are numerous factors which have to be taken into consideration when undertaking a CFD study like this, all of them will be discussed later more profoundly.

throughout this section and specifically after clearing up the main concepts of a Computational Flow Dynamics study, every aspect of the simulations and the setting of its conditions will be explained in a way that some of the work done can be seen. This will clarify which will be the conditions at which the simulation is carried out and why they have been selected or done in that way. This will lead to the final presentation of the results and its validation.

However, before entering in the explanation of specific concepts a brief summary of a few fundamental or introductory CFD notions will be presented next, in order to build a framework for what is to come.

8.1 Fundamental concepts

8.1.1 What is CFD?

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. Some examples are [21]:

- Aerodynamics of aircraft and vehicles: lift and drag.
- Hydrodynamics of ships.
- Power plant: combustion in IC engines and gas turbines.
- Turbo machinery: flows inside rotating passages, diffusers, etc.
- Electrical and electronic engineering: cooling of equipment including microcircuits.
- Chemical process engineering: mixing and separation, polymer moulding.
● External and internal environment of buildings: wind loading and heating/ventilation.
● Marine engineering: loads on off-shore structures.
● Environmental engineering: distribution of pollutants and effluents.
● Hydrology and oceanography: flows in rivers, estuaries, oceans.
● Meteorology: weather prediction.
● Biomedical engineering: blood flows through arteries and veins.

Increasingly CFD is becoming a vital component in the design of industrial processes, aside from other reasons, this is due to the availability of affordable high performance computing hardware and the introduction of user-friendly interfaces.

8.1.2 How does a CFD code work?

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: (1) a pre-processor, (2) a solver and (3) a post-processor. The function of each of these elements within the context of a CFD code will be briefly examined.

Pre-processor

Preprocessing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve:

● Definition of the geometry of the region of interest: the computational domain.
● Grid generation, that is the sub-division of the domain into a number of smaller, non overlapping sub-domains: a grid (or a mesh) of cells (control volumes or elements).
● Selection of the physical and chemical phenomena that need to be modelled.
● Definition of fluid properties.
● Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary.
8.1 Fundamental concepts

The solution to a flow problem (velocity, pressure, temperature, etc.) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells in the grid. In general the larger the number of cells the better the solution accuracy. Both the accuracy of a solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the mesh. This issue will be dealt specifically in §8.1.6.

Solver

There are three different streams of numerical solution techniques: finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps:

- Approximation of the unknown flow variables by means of simple functions.
- Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

The main differences between the three separate streams are associated with the way in which the flow variables are approximated and with the discretisation processes.

Finite difference methods. The unknowns $\phi$ of the flow problem are described by means of point samples at the node points of a grid of co-ordinate lines. Truncated Taylor series expansions are often used to generate finite difference approximations of derivatives of $\phi$ in terms of point samples of $\phi$ at each grid point and its immediate neighbours, yielding an algebraic equation for the values of $\phi$ at each grid point.

Finite element method. It uses simple piecewise functions (e.g. linear or quadratic) valid on elements to describe the local variations of unknown flow variables $\phi$. If these functions for $\phi$ are substituted into the equation it will not hold exactly and a residual is defined to measure the errors. Next these residuals (and hence the errors) are minimised in some sense by multiplying them by a set of weighting functions and integrating. As a result a set of algebraic functions for the unknown coefficients of the approximating functions are obtained.

Spectral methods. Spectral methods approximate the unknowns by means of truncated Fourier series or series of Chebyshev polynomials. Unlike the finite difference or finite element approach the approximations are not local but valid throughout the entire computational domain.
The finite volume method. It was originally developed as a special finite difference formulation. This is the most well-established and thoroughly validated general purpose CFD technique, since it is central to the majority of commercially available CFD codes including FLUENT, which is the one being used in this study. The numerical algorithm of this method consists of the following steps:

- Formal integration of the governing equations of fluid flow over all the (finite) control volumes of the solution domain.
- Discretisation involves the substitution of a variety of finite-difference-type approximations for the terms in the integrated equation representing flow processes such as convection, diffusion and sources. This converts the integral equations into a system of algebraic equations.
- Solution of the algebraic equations by an iterative method.

The most popular solution procedures (iterative methods) are the TDMA line-by-line solver and the SIMPLE algorithm to ensure correct linkage between pressure and velocity.

Post-processor

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. Owing to the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualisation tools. These include:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- View manipulation (translation, rotation, scaling etc.)
- Colour postscript output

More recently these facilities may also include animation for dynamic result display and in addition to graphics all codes produce trusty alphanumeric output and have data export
8.1 Fundamental concepts

facilities for further manipulation external to the code. As in many other branches of CAE (Computer Aided Engineering) the graphics output capabilities of CFD codes have revolutionised the communication of ideas to the non-specialist.

8.1.3 Turbulence

In 1883, experiments by physicist Osborne Reynolds on flow in a pipe showed that the behaviour of the flow was characterized by a dimensionless parameter, which was later introduced in the fluid mechanics field as the Reynolds number. This Reynolds number is a measure of the ratio of inertia forces to viscous forces.

\[ Re = \frac{\text{inertia force}}{\text{viscous force}} = \frac{\rho u^2 L^2}{\mu uL} = \frac{\rho uL}{\mu} \]  

(8.1)

Where \( L \) is the system length scale.

All flows encountered in engineering practice, both simple ones such as two dimensional jets, wakes, pipes flows and flat plate boundary layers and more complicated three dimensional ones, become unstable above a certain Reynolds number. At low Reynolds numbers flows are laminar. At higher Reynolds numbers flows are observed to become turbulent. A chaotic and random state of motion develops in which both the velocity and pressure change continuously with time within substantial regions of flow. Due to this phenomenon, a classification of the fluid flow can be made:

- Laminar flow (for \( Re \leq 2000 \)): The flow is smooth and fluid layers past each other in an orderly way, as in parallel lines.

- Transitional state (for \( 2000 \leq Re \leq 4000 \)): It is a mixture of laminar and turbulent flow.

- Turbulent flow (for \( Re \geq 4000 \)): The fluid flows in a very irregular way, changing its direction erratically. The flow is unpredictable due to the apparition of eddies.

The values that are presented as a transitional Reynolds number are only a reference, they can vary depending on the case. In the figure 8.1 an schematic representation of the experiment carried out by Reynolds is shown. Two different states of the fluid flow can be easily determined, although the third transitional state is ignored in this schematic.
Some of the characteristics of the turbulent flow are the following:

- Highly unstable.
- Diffusion: there is a rapid process of mixing of the swirling eddies of the fluid.
- Three dimensional
- Dissipation: Due to the action of viscosity the kinetic energy of the flow reduces and it is converted into internal energy.
- High Reynolds number.
- Vorticity: the vortex stretching mechanism is one of the principal mechanisms by which the intensity of turbulence is increased.

The problem of turbulence is one of the most intriguing and important problems in all classical physics. During the 19\textsuperscript{th} and 20\textsuperscript{th} Centuries, this problem has been studied by many physicists and engineers. But we do not understand yet how and why turbulence occurs. That is the reason why a prediction of the turbulent behaviour cannot be done with reliability. Turbulence is a subject on which still studies are going on.

A laminar flow can be transformed in a turbulent flow as the Reynolds number increases. This happens because small disturbances to the flow are no longer damped by the flow, they begin to grow by taking energy from the original laminar flow. Instability in the flow create eddies (fig. 8.2).
8.1 Fundamental concepts

Figure 8.2: Creation of eddies behind a cylinder.

In a turbulent flow the fluid is stirred and produces large eddies. These large eddies are unstable and interact generating smaller eddies. This smaller eddies break into even more smaller eddies, and so on. This is known as energy cascade. This energy cascade continues until the Reynolds number is sufficiently small that the eddy motion is stable, and molecular viscosity is effective in dissipating the kinetic energy.

Turbulence can be considered to consist of eddies of different sizes, these eddies are a turbulent motion localized over a region of size $l$, with a velocity $u(l)$ and a time-scale $\tau(l) = \frac{l}{u(l)}$. At eddies with a large Reynolds number the direct effects of viscosity are negligibly small. Nevertheless these eddies can become so small that molecular diffusion becomes important and viscous dissipation of energy takes place.

The scale at which this happens is the Kolmogorov length scale. The scales in the energy cascade are uncontrollable, but a division into three categories based on these length scales can be done [8]:

- Integral length scale:
  
  This is the largest scales in the energy spectrum. The eddies there obtain the energy from the mean flow and from each other. They are the eddies which contain the most of the energy, they have a large velocity fluctuation and a low frequency. The length scale of this eddies, $l_0$, is comparable to the flow length scale $L$.

- Kolmogorov length scales:

  These are the smallest scales and form the viscous sub-layer of the boundary layer (which will be explained in 8.1.5). Kolmogorov, in 1941, introduce the idea that the smallest scales of turbulence are similar for every turbulent flow and depend only on the energy dissipation $\epsilon$ and the kinematic viscosity $\nu$. Finally, at the Kolmogorov length scales, the turbulence kinetic energy is dissipated into heat through the action of molecular viscosity.
Taylor micro scales:

These are the intermediate scales between the largest and the smallest scales. In these micro scales there is no dissipation, the energy passes from the largest to the smallest without dissipation.

The studies of the Kolmogorov theory and hypothesis, its length scales and how small do eddies get are quite complicated. Since this is not the aim of this study, it stays out of the region of interest for the project, nonetheless the way the turbulence can be modelled in CFD is of great relevance, therefore the diverse models of turbulence which are used most frequently will be explained next.

8.1.4 Turbulence models

A turbulence model is a computational procedure to the system of mean flow governing equations so that a more or less wide variety of flow problems can be calculated. For most engineering purposes it is unnecessary to resolve the details of the turbulent fluctuations. Only the effects of the turbulence on the mean flow are usually sought. For a turbulence model to be useful in a general purpose CFD code it must have wide applicability, be accurate, simple and economical to run. The most common turbulence models are classified in Table 8.1.

<table>
<thead>
<tr>
<th>Classical models</th>
<th>Based on (time-averaged) Reynolds equations</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.Zero equation model</td>
<td>Algebraic model</td>
</tr>
<tr>
<td>2.One equation model</td>
<td>Prandtl model</td>
</tr>
<tr>
<td>3.Two equation model</td>
<td>$k - \epsilon$ and $k - \omega$ models</td>
</tr>
<tr>
<td>4.Reynolds stress equation model</td>
<td></td>
</tr>
</tbody>
</table>

| Large eddy simulation     | Based on space-filtered equations           |
| Direct numerical simulation| Navier-Stokes equations are numerically solved using a computer |

Table 8.1: Turbulence models

These turbulence models are used to predict the effects of turbulence in fluid flow without resolving all scales of the smallest turbulent fluctuations. Regarding the range of length and time scales that are modelled and resolved by these models, they can be classified as shown in the figure 8.3. It is remarkable that it is very difficult or even impossible to carry out a model that embrace in a precise manner the turbulence phenomenon, this is due to the wide range of length and time scales associated with turbulent flow and the randomness in time that is
8.1 Fundamental concepts

associated with turbulence. Therefore no deterministic approach is possible. However, certain properties could be learned using statistical methods, on which some of the models are based.

![Diagram illustrating the extend of modeling for certain types of turbulent models.]

**Figure 8.3:** Extend of modeling for certain types of turbulent models.

**Direct numerical simulation (DNS)**

Although not being properly a turbulence model, some aspects of this method will be considered here since it represents a way in which some difficult fluid mechanics problems could be resolved in a short-term future. In a DNS the Navier-Stokes equations are numerically solved using a computer. Obviously, this is the most accurate approach to turbulence simulation, since it does not need an averaging or approximation. In this simulation, all of the motions contained in the flow are resolved. Nonetheless, current computers are not sufficiently large and fast to permit the necessary resolution if Re is high or the problem is too complicated, hence why this method is used for small Reynolds numbers.

It can be demonstrated that the number of operations grows as $Re^3$. Nevertheless it is thought that if computers continue developing as fast as they had been doing since now, as commented earlier, in some years they could be able to solve difficult problems by using the direct numerical simulation.

Nowadays DNS can be understood as a research tool more than as a design tool. Some of the applications that can be carried out thanks to the DNS are the following:
• Understanding the mechanisms of turbulence production, energy transfer, and dissipations in turbulent flows.

• Simulation of the production of aerodynamics noise.

• Understanding the effects of compressibility on turbulence.

• Understanding the interaction between combustion and turbulence.

• Controlling and reducing drag on a solid surface.

The DNS solutions are also useful in developing turbulence models, such as in LES models.

Large Eddy Simulation (LES)

This method emerges from consideration of the fact that large eddies are flow-dependent and the small scales are more universal and independent from what is happening on the larger scales. Making this assumption the large eddies are computed accurately and the small ones are modelled. It was argued earlier that the largest eddies interact strongly with the mean flow and contain most of the energy so this approach results in a good model of the main effects of turbulence.

As said earlier, DNS gives the best results, but requires high computational costs, this is the reason why doing a separation of the eddies and using a LES model can reduce the computational costs without highly decreasing the quality in results.

These models are suitable in cases where:

• The flow is likely to be unstable, with large scale flapping of a shear layer or vortex shedding.

• The flow is likely to be unsteady with coherent structures.

• The flow is buoyant, with large unstable regions created by heating from below, or by lighter fluid below heavier fluid.

• Conventional RANS approach are known to fail.

• A good representation of the turbulent structure is required for small-scale processes such as micro-mixing or chemical reaction.
8.1 Fundamental concepts

• The noise from the flow is to be calculated, and especially when the broadband contribution is significant.

• Other fluctuating information is required.

The specific procedure and filtered equations through which the large eddies are computed will not be discussed further since it is out of the scope of this project.

Reynolds-averaged Navier-Stokes (RANS)

The idea of this method is to use the Reynolds decomposition to decompose a function into a time-averaged part and a fluctuating part. Using the Reynolds decomposition, a function \( \tilde{u}(x, t) \) can be expressed as:

\[
\tilde{u}(x, t) = U(x) + u(x, t)
\] (8.2)

Where \( u(x, t) \) is the fluctuating part, and \( U(x) \) is the steady mean value independent of time.

![Plot of the Reynolds decomposition](image)

Considering now the Navier-Stokes equations for incompressible flows (\( \rho = cte \)) and without the body-force term:

\[
\nabla \cdot \tilde{u} = 0
\] (8.3)

\[
\rho \left( \frac{\partial \tilde{u}}{\partial t} + \tilde{u} \cdot \nabla \tilde{u} \right) = -\nabla p + \mu \cdot \Delta \tilde{u}
\] (8.4)

And applying the Reynolds decomposition and eq. 8.3, it can be obtained:

\[
\rho \left( \frac{\partial}{\partial x_j} (U_i U_j) + \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) \right) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i}
\] (8.5)

Where \( \tau_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \) is the mean viscous tensor and \( \rho \bar{u}_i \bar{u}_j \) is the Reynolds stress tensor.
The appearance of this tensor is what makes the problem so difficult to solve. The Reynolds stress is a property of the flow so it is dependent on the flow variables themselves. This stress changes from flow to flow and no general relations are available.

The objective of the turbulence models for the RANS equations is to compute the Reynolds stress, which can be done by three main categories of RANS-based turbulence models: the linear eddy viscosity models, the nonlinear eddy viscosity models or through Reynolds stress models (RSM).

**Linear eddy viscosity models**

In these models the Boussinesq hypothesis, which assumes that the Reynolds stress is proportional to the mean rate of strain, is used:

\[- \rho \overline{u_i u_j} = 2 \mu_t S_{ij} - \frac{2}{3} \rho k \delta_{ij} \]  

(8.6)

Where the mean strain rate is described as $S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{1}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right)$ and $k$ is the turbulent kinetic energy.

Once this assumption is made, the eddy viscosity, $\mu_t$, has to be modelled. Depending on the number of equations solved to compute the eddy viscosity coefficient a classification of the different eddy viscosity models can be done:

- Zero equation models.
- One equation models.
- Two equation models.
8.1 Fundamental concepts

**Zero equation models**

The zero equation models or algebraic models are the ones that do not require the solution of any additional equations. They compute a global value for the turbulence viscosity from the mean velocity and geometric length scale using an empirical formula. As a consequence, zero equation models may not be able to properly account for history effects on the turbulence, such as convection and diffusion of turbulent energy.

\[ \mu_t = \rho f_\mu U_t l_t \]  

Where \( f_\mu = 0.01 \) is a constant and \( l_t = \frac{V_D^\frac{3}{2}}{\bar{U}} \), where \( V_D \) is the fluid domain volume.

These equations are simple to implement and use, and can produce approximate results very quickly. These results are often too simple for use in general situations, but they provide a good initial guess for simulations using more advanced turbulence models.

**One equation models**

One equation turbulence models solve one turbulent transport equation to obtain the value of the eddy viscosity. Prandtl, in 1945, calculated the turbulent viscosity as a function of turbulent kinetic energy \( k \). One advantage of this model is that it takes into account the flow history.

The original Prandtl model is:

\[ \frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_D \frac{k^{\frac{3}{2}}}{l} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_j} \right] \]  

Where \( \nu_t = \frac{\mu_t}{\rho} = k^{\frac{3}{2}} l = C_D \frac{k^2}{\epsilon} \) is the kinematic eddy viscosity, \( C_D \) is a constant with values between 0.7 and 0.9 and \( \sigma_k = 1 \).

The original one-equation model is Prandtl’s one-equation model, which has just been presented. Other common one-equation models are [10]:

- Baldwin-Barth model
- Spalart-Allmaras model
- Rahman-Agarwal-Siikonen model
The Spalart-Allmaras model is relatively simple and is well suited for aerospace applications (wall-bounded flows) and provides good results for boundary layers subjected to adverse pressure gradients. This one is taken into consideration for the final selection of the model.

Two equation models

Two equation models are one of the most common types of turbulence models. They are used in most of the engineering problems. These models include two extra transport equations to represent the turbulent properties of the flow. Using this model only requires introducing the initial and boundary conditions; there is no necessity to have some knowledge about the studied flow in advance.

- \( k - \epsilon \) models

The transported variables in this case are the kinetic energy \( k \) and the turbulent dissipation \( \epsilon \). In this model the eddy viscosity is obtained as:

\[
\mu_t = \rho C_\mu \frac{k^2}{\epsilon}
\]  

(8.9)

Where \( C_\mu = 0.09 \)

There are different models to obtain the transport variables \( k \) and \( \epsilon \).

In the Standard \( k - \epsilon \) model the eddy viscosity is determined from a single turbulence length scale, so the calculated turbulent diffusion is that which occurs only at the specified scale, whereas in reality all scales of motion will contribute to the turbulent diffusion.

The RNG \( k - \epsilon \) model takes into account the smaller scales of motion by changing the constants of the Standard \( k - \epsilon \) model, but using the same equations for \( k \) and \( \epsilon \). In general, turbulence models based on the \( \epsilon \)-equation predicts the onset of separation too late and under predicts the amount of separation later on. These models are not suitable for:

- Flows with boundary layer separation.
- Flows with sudden changes in the mean strain rate
- Flows in rotating fluids
8.1 Fundamental concepts

- Flows over curved surfaces

• $k - \omega$ models

The transported variables in this case are the kinetic energy $k$ and the specific dissipation $\omega$. One of the main problems in turbulence modeling is the accurate prediction of flow separation from a smooth surface. The $k - \omega$ models try to avoid the problems of the $k - \epsilon$ models. The advantage of the $k - \omega$ formulation is the near wall treatment for low-Reynolds number.

Using these models the eddy viscosity is obtained as:

$$\mu_t = \rho \frac{k}{\omega} \quad (8.10)$$

Taking into account the different regions of a boundary layer, which will be explained in 8.1.5, a comparison between the $k - \omega$ models and the $k - \epsilon$ models can be done as follows:

<table>
<thead>
<tr>
<th>Model</th>
<th>$k - \omega$</th>
<th>$k - \epsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sublayer</td>
<td>Robust</td>
<td>Stiff</td>
</tr>
<tr>
<td></td>
<td>Simple</td>
<td>Less accurate</td>
</tr>
<tr>
<td></td>
<td>Accurate</td>
<td>Complex</td>
</tr>
<tr>
<td>log-layer</td>
<td>Accurate</td>
<td>Large length scales</td>
</tr>
<tr>
<td>Wake region</td>
<td>Missing transport effects</td>
<td>Missing transport effects</td>
</tr>
<tr>
<td>Boundary layer edge</td>
<td>Free-stream sensitive</td>
<td>Well defined</td>
</tr>
</tbody>
</table>

Table 8.2: Comparison between $k - \omega$ and $k - \epsilon$ models.

Seeing the table 8.1 it is clear that the $k - \omega$ models have a higher quality in the inner part of the boundary layer and they give worst results as we move away from it. On the other hand, the $k - \epsilon$ models have a higher quality in the outer part of the boundary layer and they reduce their accuracy as they get closer to the inner part of the boundary layer.

The Shear stress transport $k - \omega$ model (SST $k - \omega$) uses this knowledge to obtain high quality results combining the use of both models. In the inner parts of the boundary layer a $k - \omega$ formulation is used whereas it switches to a $k - \epsilon$ behaviour in the free-stream.
avoiding the common $k - \omega$ sensitivity problem in this region [10]. This model gives a high accurate prediction of the onset.

The eddy viscosity is computed as:

$$\mu_t = \frac{a_1 k \rho}{\max(a_1 \omega, SF_2)}$$ (8.11)

Where $F_2$ is a blending function which restricts the limiter to the wall boundary layer:

$$F_2 = \tanh \left( \max \left( \frac{2 \sqrt{K} \cdot 500 \nu}{\beta' \omega y' y^2} \right) \right)^2$$ (8.12)

$$S = \sqrt{2S_{ij} S_{ij}}$$ (8.13)

$$S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$ (8.14)

$$a_1 = 0.31$$ (8.15)

This model was developed to overcome deficiencies in the other models, that is why it is recommended to use the SST model over others $k - \omega$ models or $k - \epsilon$ models. This is one of the options allowed by the FLUENT software, the one that is used in this study, and according to the literature is one of the most accurate models, it implies a certain expense regarding the computing power required. In order to optimize the resources provided, the model that will be used in this study is the Spalart-Allmaras model for its simplicity and good results in external turbulent flows, this was decided after some simulations done with the SST-$k - \omega$ model, where it could be seen that perhaps this model was too complex for a such case of a solar panel.

**Nonlinear eddy viscosity models**

This is a turbulence model for the RANS equations in which an eddy viscosity coefficient is used to relate the mean turbulence field to the mean velocity field, however in a nonlinear relationship.

$$- \rho(u_i u_j) = 2 \mu_t F_{nl}(S_{ij} \Omega_{ij}, ...)$$ (8.16)
8.1 Fundamental concepts

Where

- $F_{nl}$ is a nonlinear function possibly dependent on the mean strain and vorticity fields or even other turbulence variable.

- $\mu_t$ is the coefficient termed turbulence "viscosity" (also called eddy viscosity).

- $S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij}$ is the mean strain rate.

- $\Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right)$ is the mean vorticity.

These models are applied in flows where the turbulent transport or non-equilibrium effects are important, that is when the eddy-viscosity assumption is no longer valid.

**Reynolds Stress models (RSM)**

The Reynold’s Stress Models (RSM), also known as the Reynold’s Stress Transport (RST) models, are higher level, elaborate turbulence models. The method of closure employed is usually called a Second Order Closure. This modelling approach originates from the work by Launder (1975). In RSM, the eddy viscosity approach has been discarded and the Reynolds stresses are directly computed. The exact Reynolds stress transport equation accounts for the directional effects of the Reynolds stress fields.

The calculation of the individual Reynolds stresses is done using differential transport equations. The individual Reynolds stresses are then used to obtain closure of the Reynolds-averaged momentum equation.

**Summary**

As shown in figure 8.1, DNS and LES models resolves shorter length scales than RANS models, hence better results can be obtained by using these models over the RANS models. Nevertheless, as it is expected, the more detailed solutions that are wanted to be obtained, the higher computational cost that is needed. Thus being necessary to do an evaluation between the degree of the detail desired and the computational cost that can be assumed.

For most CFD simulations used as a design tool, which is this case, there is no need of achieving the degree of detail that DNS and LES offer. Generally, RANS models results are detailed enough in such cases. Being this the main reason why RANS models are nowadays the most
widely used models, since they offer a significant degree of detail without demanding higher computational cost.

Of course there are models which have not been explained in this section, this is the case of some hybrid models, like the DES (Detached eddy simulation) model, which combines RANS and LES methods and is used for analysis for flows around non aerodynamic shapes. However what has been explained here is intended to clear the concept of the turbulence models in CFD and also focusing and define the one selected for the study and not to do a compilation of all the existing turbulence models. All in all and after having presented different models for the turbulence some recommendations when choosing a model can be done:

- For free shear flows of for boundary layer in equilibrium the differences between the models are fairly small.
- For flows which need a high accuracy in boundary layers a SST model should better be used.
- In the case of flows with strong swirl a Reynolds Stress model should be used.

8.1.5 Boundary layer

The boundary layer is a thin region on the surface of a body in which viscous effects are important. The effect of friction near the wall causes the fluid immediately adjacent to the surface to stick to it, this effect takes place only in a thin region near the surface, the boundary layer. The velocity of the fluid at the surface is zero and it increases enormously within a thin layer until it reaches the free stream value of the velocity away from the surface. This region of very large velocity gradients is the boundary layer [5].

There are laminar boundary layers and turbulent boundary layers depending on the Reynolds number.

Laminar boundary layer flow

The laminar boundary is a very smooth flow, while the turbulent boundary layer contains swirls or "eddies." The laminar flow creates less skin friction drag than the turbulent flow, but is less stable. Boundary layer flow over a wing alike surface begins as a smooth laminar flow. As the flow continues back from the leading edge, the laminar boundary layer increases in thickness.
Turbulent boundary layer flow

At some distance back from the leading edge, the smooth laminar flow breaks down and transitions to a turbulent flow. From a drag standpoint, it is advisable to have the transition from laminar to turbulent flow as far aft on the aerodynamic surface as possible, or have a large amount of the surface within the laminar portion of the boundary layer. The low energy laminar flow, however, tends to break down more suddenly than the turbulent layer.

The concept of the boundary layer was first introduced by Ludwig Prandtl at the third Congress of Mathematicians at Heidelberg, Germany, in 1904. The below quotation is taken from this historic paper:

*A very satisfactory explanation of the physical process in the boundary layer between a fluid and a solid body could be obtained by the hypothesis as an adhesion of the fluid to the walls, that is by the hypothesis of a zero relative velocity between fluid and wall. If the viscosity was very small and the fluid path along the wall not too long, the fluid velocity ought to resume its normal value at a very short distance from the wall. In the thin transition layer however, the sharp changes in velocity, even with small coefficient of friction, produce marked results.*

Ludwig Prandtl, 1904

This concept eventually revolutionized the analysis of viscous flows in the twentieth century and allowed the practical calculation of drag and flow separation over aerodynamic bodies. Before Prandtl’s 1904 paper, the Navier-Stokes equations were well known, but fluid dynamicists were frustrated in their attempts to solve these equations for practical engineering problems.
After 1904, using Prandtl’s concept of a boundary layer adjacent to an aerodynamic surface, the Navier-Stokes equations can be reduced to a more tractable form called the boundary layer equations. These equations can be solved to obtain the distributions of shear stress and aerodynamic heat transfer to the surface.

**Boundary layer equations**

In the boundary layer occurs that:

- The diffusive transport of momentum in the principal flow direction is much smaller than convection, and so, it can be neglected.
- The velocity component in the main flow direction is much larger than the components in other directions.
- The pressure gradient across the flow is much smaller than in the principal flow direction.

Therefore, the Navier-Stokes equations can be simplified for a steady two dimensional, laminar and incompressible flow as follows:

\[
\frac{u}{\partial x} + \frac{v}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{\rho} \frac{\partial^2 u}{\partial y^2} \quad (8.17)
\]

With the continuity equation being:

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (8.18)
\]

The boundary conditions to be imposed are, on the surface:

\[
y = 0, \quad u = 0, \quad v = 0
\]

On the outer edge of the boundary layer:

\[
y = \delta, \quad u = U(x)
\]

Being \(\delta\) the boundary layer thickness, which is defined as the distance, starting from the surface, that the flow needs to nearly achieve the free stream velocity \(u_0\). It is said that the boundary layer ends when the velocity of the flow reaches the 99% of the free stream velocity.

However, these equations refers to the most simple case, and specifically in the case of study which will be developed in this report, the boundary layer will tend to be turbulent in most
8.1 Fundamental concepts

of the surfaces, because of the geometry of the object, which does not present totally flat and smooth surfaces.

Boundary sub-layers

The boundary layer can be divided in four sub-layers [1].

1. Linear or viscous sub-layer:
   At the solid surface the fluid is stationary. As there are no turbulent Reynolds shear stress effects, the viscous stresses dominate the flow adjacent to the surface and so friction must be taken into account. This layer is extremely thin \((y^+ < 5)\) and it is often assumed that the shear stress is approximately constant and equal to the wall shear stress. It can finally be demonstrated that \(u^+ = y^+\).

   Where \(y^+\) is the dimensionless distance from the wall \(y^+ = y \frac{u_w}{\sqrt{\nu}}\), \(u^+ = \frac{u(y)}{u_r}\), \(u_r = \sqrt{\frac{\tau_w}{\rho}}\), being \(\tau_w\) the wall shear stress and \(y\) the distance from the wall.

2. Buffer layer:
   This layer connects the viscous sub-layer to the inertial sub-layer. In this layer, inertial and dissipation effects are nearly balanced. This layer goes from \(y^+ > 5\) to \(y^+ < 30\).

3. Inertial or log-law layer:
   In this layer the turbulent Reynolds stresses dominate the flow. The velocity follows the log-law:

   \[
   u^+ = \frac{1}{k} \ln(y^+) + B
   \]

   \(8.19\)
Where $k$ is the von Kármán constant, $k = 0.41$, and $B$ is an empirical constant with a value of $B = 5.5$ in the case of smooth walls, this value decrease in the case of wall roughness.

This layer goes from $y^+ = 30$ to $y^+ = 500$.

4. **Outer or defect layer:**

For larger values of $y$, the wake contribution needs to be taken into account, so the mean velocity profile over the whole boundary layer is well predicted by the sum of the law of the wall and the law of the wake. The correct law in this region is the velocity-defect law:

\[
U^+ = \frac{1}{k} \ln \left( \frac{y}{\delta} \right) + \frac{\Pi}{k} \left( W \left( \frac{y}{\delta} - 2 \right) \right)
\]

(8.20)

Where $\Pi$ is the wake strength parameter and is flow dependent and $W \left( \frac{y}{\delta} \right)$ is the wake function which is supposed to be universal and a convenient approximation for it is $W \left( \frac{y}{\delta} \right) = 2 \sin^2 \left( \frac{\pi y}{2 \delta} \right)$

In this region, the viscosity can be neglected, and the flow corresponds to the inviscid limiting solution.

**Separation of the boundary layer**

Flow separation occurs when the boundary layer travels far enough against an adverse pressure gradient that the speed of the boundary layer relative to the object falls almost to zero. The fluid flow becomes detached from the surface of the object, and instead takes the forms of eddies and vortices.

The position of separation is given by the condition that the velocity gradient perpendicular to the wall vanishes at the wall, that means, the wall shear stress vanishes:

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

(8.21)
8.1 Fundamental concepts

![Figure 8.7: Boundary layer separation.](image)

Near-wall modelling

For the case of study that is portrayed here, a turbulent boundary layer is expected, due to the presence of sharp edges and discontinuities in the geometry and its flat plate alike shape. It is known that in turbulent flows the velocity fluctuations within the turbulent boundary layer can be a significant percentage of the mean flow velocity, so it is critical that these effects are captured with accuracy. When it comes to the treatment of the boundary layer, ANSYS FLUENT presents two main options for the near-wall modelling strategy [14] [15]:

1. **Resolving the Viscous sub-layer**

   In the laminar sub-layer region \((y^+ < 5)\) inertial forces are less domineering and the flow exhibits laminar characteristics, which is why this is known as the low-Re region. Low-Re turbulent models (e.g. the SST model) aim to resolve this area and therefore require an appropriate mesh resolution to do this with accuracy. This is most critical for flows with a changing pressure gradient where we expect to see separation, as observed below.
Here some specific traits, that should be taken into account when using this approach, are presented:

- Involves the full resolution of the boundary layer and is required where wall-bounded effects are of high priority (adverse pressure gradients, aerodynamic drag, pressure drop, heat transfer, etc.)
- Wall adjacent grid height must be order $y^+ = O$ (The first grid point must be placed within the viscous sub-layer or near-wall region, approximately at $y^+ < 5$).
- Must use an appropriate low-Re number turbulence model (i.e. Shear Stress Transport, Spalart-Allmaras).

2. Adopting a Wall Function Grid

It is possible to use semi-empirical expressions known as wall functions to bridge the viscosity-affected region between the wall and the fully-turbulent region. The main benefit of this wall function approach lies in the significant reduction in mesh resolution and thus reduction in simulation time. However, the shortcoming lies in numerical results deteriorating under subsequent refinement of the grid in wall normal direction (thus reducing the $y^+$ value into the buffer layer zone). Continued reduction of $y^+$ to below 15 can gradually result in unbounded errors in wall shear stress and wall heat transfer (due to the damping functions inherent within the wall function approach).

Some aspects to be taken into consideration with this approach:

- Involves modelling the boundary layer using a log-law wall function. This approach is suitable for cases where wall-bounded effects are secondary, or the flow undergoes...
8.1 Fundamental concepts

geometry-induced separation (such as many bluff bodies and in modern automotive vehicle design).

- Wall adjacent grid height should ideally reside in the log-law region where $y^+ > 11$.
- All turbulence models are applicable (e.g. Shear Stress Transport or k-$\epsilon$ with scalable wall functions)

From the last paragraphs it is remarkable how the $y^+$ from the first grid point (the wall adjacent cell centre) to the wall, is so important when it comes to decide which approach to use when dealing with the boundary layer. Since here the geometry can not be assimilated as being a bluff or a bulky body and the aerodynamic drag is wanted to be calculated with considerable accuracy, attention will be paid in this aspect, this translates in checking if the cell wall distance of the mesh $y^+$ is the desired for the turbulence model for every FLUENT case.

8.1.6 Meshing

The partial differential equations that govern fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyze fluid flows, flow domains are split into smaller subdomains (made up of geometric primitives like hexahedra, tetrahedra and prisms in 3D and quadrilaterals and triangles in 2D), which are often called elements or cells, and the collection of all elements or cells is called a mesh or grid. The governing equations are then discretized and solved inside each of these subdomains.

![Figure 8.9: Elements used in a mesh or grid.](image)

Care must be taken to ensure proper continuity of solution across the common interfaces between two subdomains, so that the approximate solutions inside various portions can be put together to give a complete picture of fluid flow in the entire domain. The choice of a mesh has a significant impact on the solution accuracy, the rate of convergence and the time needed to obtain the solution.

The process of obtaining an appropriate mesh (or grid) is termed mesh generation (or grid
generation), and has long been considered a bottleneck in the analysis process due to the lack of a fully automatic mesh generation procedure. Specialized software programs have been developed for the purpose of mesh and grid generation, and access to a good software package and expertise in using this software are vital to the success of a modelling effort. In this study the software that will be used for this purpose is ANSYS ICEM CFD, a popular proprietary software package which provides advanced geometry acquisition, mesh generation, mesh optimization and post-processing tools.

Mesh classification

The most basic form of mesh classification is based upon the connectivity of the mesh: structured, unstructured or hybrid \[6\] \[11\].

1. **Structured meshes**
   A structured mesh is characterized by regular connectivity that can be expressed as a two or three dimensional array. This restricts the element choices to quadrilaterals in 2D or hexahedra in 3D. The regularity of the connectivity allows us to conserve space since neighbourhood relationships are defined by the storage arrangement. the positions of the nodes of a face for example, can be stored in two-dimensional arrays, so that it demands less computational memory than an unstructured mesh.

2. **Unstructured meshes**
   An unstructured mesh is characterized by irregular connectivity. There is no relationship between the indices of one node and the indices of the one next to it. A separate list containing the connectivity information of the nodes has to be stored, that is why the storage requirements for this type of meshes are larger than in structured meshes. However, most commercial CFD software (like ANSYS FLUENT) uses only the unstructured storage type, also for structured meshes.

![Structured and unstructured mesh discretization.](image)

3. **Hybrid meshes**
8.1 Fundamental concepts

A hybrid grid contains a mixture of structured portions and unstructured portions. It integrates the structured meshes and the unstructured meshes in an efficient manner. Those parts of the geometry that are regular can have structured grids and those that are complex can have unstructured grids.

![Figure 8.11: Example of a hybrid mesh.](image)

Mesh quality

Based on different factors, the quality of the mesh can be measured and therefore the suitability of the mesh can be decided [11].

- Skewness:
  Skewness is defined as the ratio of the difference between the size of one cell and an optimal equilateral cell of equivalent volume and the optimal cell size.

\[
\text{Skewness} = \frac{\text{optimal cell size} - \text{cell size}}{\text{optimal cell size}}
\]  

(8.22)

The quality of the cell is better when the value of skewness is closer to zero, and worst as it approaches to one.

Another common measure of quality is based on equiangular skew.

\[
\text{Equiangular skew} = \max \left[ \frac{\theta_{\text{max}} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{\text{min}}}{\theta_e} \right]
\]  

(8.23)
Where:

\[ \theta_{\text{max}} \] is the largest angle in a face or cell.

\[ \theta_{\text{min}} \] is the smallest angle in a face or cell.

\[ \theta_e \] is the angle for equi-angular face or cell i.e. 60\(^\circ\) for a triangle and 90\(^\circ\) for a square.

Again, a skewness of 0 is the best possible one and a skewness of one is almost never preferred. For Hex and quad cells, skewness should not exceed 0.85 to obtain a fairly accurate solution, whereas for triangular cells, skewness should not exceed 0.85 and for quadrilateral cells, skewness should not exceed 0.9.

- **Smoothness:**
  The change in size from one element to the following one should be gradual. Sudden changes in cell size need to be avoided. Ideally, the maximum change in grid spacing should not be greater than the 20

- **Aspect ratio:**
  The aspect ratio measures the stretching of the cell. It is defined as the ratio of the longest to the shortest side in a cell. Ideally, it should be equal to 1 to ensure best results, but values until 5 are also accepted.

- **Mesh density:**
  The density of the mesh is required to be sufficiently high in order to capture all the flow features but on the same note, it should not be so high that it captures unnecessary details of the flow, thus burdening the CPU and wasting more time.

If the mesh fulfills all the mesh quality requirements, a good and reliable solution can be obtained.

**Deciding the type of mesh**

ANSYS ICEM CFD’s mesh generation tools offer the capability to parametrically create meshes from geometry in numerous formats:

- Multiblock structured
- Unstructured hexahedral
8.1 Fundamental concepts

- Unstructured tetrahedral
- Cartesian with H-grid refinement
- Hybrid meshes comprising hexahedral, tetrahedral, pyramidal and/or prismatic elements
- Quadrilateral and triangular surface meshes

If the accuracy is of the highest concern then hexahedral mesh is the most preferable one. Whenever a wall is present, the mesh adjacent to the wall is fine enough to resolve the boundary layer flow and generally quad, hex and prism cells are preferred over triangles, tetrahedrons and pyramids. Quad and Hex cells can be stretched where the flow is fully developed and one-dimensional.

Based on the latter, a structured hexahedral mesh has been decided to be used in this study. However, as it has been already mentioned, the ANSYS FLUENT software supports and reads only the unstructured type of mesh, so in order to import the mesh to FLUENT it must be saved as an unstructured one although it is in fact a structured mesh with hexahedral elements.

8.1.7 Boundary conditions

Although the governing equations are the same \( \text{6.1.2} \) for all the flows no matter the geometry or case to be studied, the flow fields are different for every case. To obtain the exact solution in every situation, boundary conditions need to be applied. The boundary conditions are quite different for each case, and they dictate the particular solutions.

Almost every computational fluid dynamics problem is defined under the limits of initial and boundary conditions. The main boundaries in a fluid problem are:

- inlet
- outlet
- wall
- symmetry plane
Inlet

In the inlet boundary the fluid predominantly flows into the domain. Depending on the Mach number we can choose between a subsonic flow and a supersonic flow. The Mach number is defined as the fluid flow velocity, $n$, divided by the speed of sound in the medium, $a$.

$$Ma = \frac{V}{a} \tag{8.24}$$

The subsonic flow is the one which has a Mach number smaller than the unity. On the other hand, the supersonic flow has a Mach number greater than one.

Subsonic flow

In the inlet boundary condition the velocity or pressure must be specified. The different options that the program ANSYS FLUENT offers are [2]:

- Normal speed: the magnitude of the resultant normal velocity at the boundary is specified. The value specified is transferred from the fluid domain normal to each element face on that boundary.

- Cartesian velocity components: specify the Cartesian components of velocity on the inlet boundary.

- Cylindrical velocity components: specify the r, theta and z components of the velocity on the inlet boundary in cylindrical coordinates.
8.1 Fundamental concepts

- Mass flow rate: the total mass flow rate into the domain at the boundary is specified.
- Total pressure: the relative total pressure and a flow direction are specified.
- Stationary frame total pressure: this is the same as the total pressure condition in a stationary domain.
- Static pressure: the relative static pressure and a flow direction are specified.
- Fluid velocity: this option is only available for an inhomogeneous multiphase simulation.

Supersonic flow

This case can only be imposed if the total energy model is employed. The fluid should be an ideal gas, a real fluid, or a general fluid whose density is a function of pressure.

Turbulence level

The turbulence level, also known as turbulence intensity is defined as:

\[ I = \frac{u'}{U} \]  

Where \( u' \) is the root-mean-square of the turbulent velocity fluctuations, and \( U \) is the mean velocity.

When setting boundary conditions for a CFD simulation it is often necessary to estimate the turbulence intensity on the inlets. To do this accurately it is good to have some form of measurements or previous experience to base the estimate on. Here are a few examples of common estimations of the incoming turbulence intensity [9]:

1. High-turbulence case: High-speed flow inside complex geometries like heat-exchangers and flow inside rotating machinery (turbines and compressors). Typically the turbulence intensity is between 5% and 20%

2. Medium-turbulence case: Flow in not-so-complex devices like large pipes, ventilation flows etc. or low speed flows (low Reynolds number). Typically the turbulence intensity is between 1% and 5%

3. Low-turbulence case: Flow originating from a fluid that stands still, like external flow across cars, submarines and aircraft. Very high-quality wind-tunnels can also reach really low turbulence levels. Typically the turbulence intensity is very low, well below 1%.
The eddy viscosity ratio or turbulent viscosity, which must be defined often, is simply the ratio of turbulent to laminar (molecular) viscosity:

\[
\beta = \frac{\nu_t}{\nu}
\]  

(8.26)

For internal flows \( \beta \) may be scaled with the Reynolds numbers, whereas for external flows the freestream turbulent viscosity will be on the order of laminar viscosity so small values of \( \beta \) are appropriate.

**Outlet**

In the outlet boundary the fluid predominantly flows out of the domain. The different options that the program ANSYS offers to define the outlet are:

- Static pressure: The relative pressure is maintained at a fixed specified value over the outlet boundary.  
- Normal speed: specify the magnitude of the flow velocity at the outlet.  
- Cartesian velocity components: the boundary velocity components are specified.  
- Cylindrical velocity components: the components and axis are specified in the same way as for an inlet.  
- Average static pressure: with this option, the static pressure is allowed to locally vary on the outlet boundary such that the average pressure is constrained in a specified manner.  
- Mass flow rate: the mass flow rate through the outlet boundary is specified.  
- Degassing condition: this option is used to model a free surface from which dispersed bubbles are permitted to escape, but the liquid phase is not.  
- Fluid velocity: this option is only available for an inhomogeneous multiphase simulation.  
- Supercritical: Supercritical free surface flow means that the liquid velocity exceeds the local wave velocity, and nothing needs to be set at the outlet.

There are recommended configurations of boundary conditions due to the robustness of the problem:
8.1 Fundamental concepts

<table>
<thead>
<tr>
<th></th>
<th>Inlet</th>
<th>Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Most robust</td>
<td>Velocity</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Robust</td>
<td>Total pressure</td>
<td>Velocity</td>
</tr>
<tr>
<td>Sensitive to initial guess</td>
<td>Total pressure</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Very unreliable</td>
<td>Static pressure</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Not possible</td>
<td>Static pressure</td>
<td>Total pressure</td>
</tr>
</tbody>
</table>

Table 8.3: Recommended configurations for boundary conditions.

Opening

When having an opening boundary condition the flow is into and/or out of the domain. The fluid flows in both directions across the boundary.

The different methods of defining an opening are:

- Cartesian velocity components
- Cylindrical velocity components
- Opening pressure and direction
- Static pressure and direction
- Entrainment: this option is useful for situations in which the main flow tends to pull fluid through the boundary where the flow direction is unknown
- Fluid velocity

Wall

A wall is an impenetrable boundary to fluid flow. Walls allow the permeation of heat and additional variables into and out of the domain through the setting of flux and fixed value conditions at wall boundaries.

The different types of defining a wall are:

- No slip wall: In the case of viscous fluids there is no relative velocity between the surface and the fluid immediately at the surface. The fluid next to the wall assumes the velocity of the wall.
- Free slip wall: For an inviscid fluid, the flow slips over the surface as there is no friction. The shear stress at the wall is zero, and the velocity of the fluid near the wall is not retarded by wall friction effects.

- Finite slip wall: The fluid slips at the wall when the wall shear stress is greater than a critical stress. (Typical use to simulate the flow of a non-Newtonian fluid)

- Counter-rotating wall: The wall boundary is assumed to be stationary.

- Rotating wall: Enables the wall to rotate with a specified angular velocity.

Symmetry plane

The symmetry plane is used to define a symmetric problem about a plane when the flow on one side of the plane is a mirror image of flow on the opposite side.

8.2 CFD study

To get to a final CFD solution different steps had to be followed:

8.2.1 Creation of the geometry

First of all, the geometry has to be defined. In order to do that, models of the different configurations of solar modules corresponding to each solar tracker have been created using the SolidWorks 3D CAD design program, which were then imported in the ANSYS workbench. It must be said that the author was provided with models already done for every configuration, nonetheless, these models were very detailed and thus not appropriated for the creation of the mesh. For this reason, it was decided to simplify the geometry of the models.

Report - 62
The fluid flow volume or domain must be defined also, in order to get to study the fluid flow around the solar modules, for doing so the dimensions of the fluid flow has been defined to be...
the following, based on [13] and represented in the figure 8.16. In the figure 8.17 the actual fluid domain is shown, before the generation of the mesh/grid.

![Figure 8.16: Representation of the fluid domain dimensions [13].](image1)

![Figure 8.17: Fluid domain around the PVAST 327 modules in ANSYS ICEM CFD.](image2)

### 8.2.2 Meshing

Once the basic geometry is defined, it is time to create the mesh, which is a tedious process. In this study the mesh was created using the ICEM CFD module of ANSYS software. Since it had already been decided that the mesh would be an structured hexahedral mesh, it was created using a useful tool of ICEM, that is the blocking procedure which helps a lot in the creation of hexa mesh. Summarizing the procedure followed or what it is needed to generate a mesh with hexa is [3]:

Report - 64
8.2 CFD study

- Import of a geometry file using any of the direct, indirect or faceted data interfaces. In this specific case the geometry of the solar modules was saved as .STEP files in SolidWorks and imported to ICEM CFD.

- Interactively define the block model through split, merge, O-grid definition, edge/face modifications and vertex movements.

- Check the block quality to ensure that the block model meets specified quality thresholds.

- Assign edge meshing parameters such as maximum element size, initial element height at the boundaries and expansion ratios (how the cell volume is changing with respect to the neighbouring cells).

- Generate the mesh with or without projection parameters specified. Check mesh quality to ensure that specified mesh criteria are met.

- Write output files to the desired solvers, in this case FLUENT.

![Figure 8.18: Process of the blocking of the geometry.](image)
All through this process of the generation of the grid, sometimes it was mandatory to coarsen, refine or adapt the mesh according to the lack of convergence of the iterations. However, it was possible to get to an agreement point where the results of the solver could be accepted. It is remarkable the relevance of the refinement at the boundary layer zone (fact that was already commented in 8.1.5), the convergence from iterations was very sensitive to this aspect, thus special treatment is applied in this zone:

---

Figure 8.19: Final blocking of the geometry.
8.2 CFD study

8.2.3 Boundary conditions setting

Before writing the output or exporting the mesh, in order to be able to used it in FLUENT, it is necessary to set the boundary conditions of the fluid domain. To do so and according to the robustness criteria stated in table 8.12 the boundary conditions has been defined as follows:
8.2.4 Case setup

Once the mesh is ready, the next step is to set-up the case in the FLUENT graphical interface. Before doing so, it is advisable to check the mesh, although it has been already checked with ICEM CFD, the FLUENT software takes into consideration supplementary factors like the cell wall distance which is very important to the treatment of the boundary layer, as commented on 8.1.5.

General

In the General panel the density based solver is selected. Though the pressure-based approach was developed for low-speed incompressible flows (which is this case), while the density-based approach was mainly used for high-speed compressible flows, recently both methods have been extended and reformulated to solve and operate for a wide range of flow conditions beyond their traditional or original intent.
8.2 CFD study

![General panel of FLUENT](image1)

Figure 8.23: Settings in the *General* panel of FLUENT.

**Models**

Here the turbulence model is selected, in this case the one selected is the *Spalart-Allmaras* model, for reasons already explained in [8.1.4](#).

![Turbulent model selection](image2)

Figure 8.24: Selection of the turbulent model.

Due to the fact that this case is a external incompressible flow where no remarkable thermal effects are expected, the *Energy equation* is turned off.

**Materials**

In the Materials panel, for the air a constant density of 1.225 $kg/m^3$ is fixed, since this is a incompressible case in which the velocity is under 0.2 Mach. Viscosity is also kept as constant with the default value of 1.7894 $\cdot 10^{-5}$ $kg/m/s$, that is because no thermal effects are taken into account in this simulation, so there is no need to make the viscosity dependent of temperature.
Cell zone conditions

In this panel, specifically speaking in the Operating Conditions option, the Operating Pressure is set to the atmospheric pressure 101325 Pa, for incompressible flows it is normal to specify a large operating pressure and let the solver work with smaller 'gauge' pressures for the boundary conditions, to reduce round-off errors.

Boundary conditions

For the boundaries defined as wall or symmetry plane there is nothing more to do, whereas for the inlet and outlet some settings have to be done:

- Inlet: The magnitude of the free-stream velocity is specified and is defined to be normal to the boundary. The turbulence viscosity ratio is defined with a value of 10%.

- Outlet: The default value of the Gauge pressure 0 Pa is kept. The value for the turbulence viscosity ratio is the same as the one from the inlet.

Reference values

Normally the free-stream is used as a reference condition, here these reference values are computed from the inlet.

Solution methods
8.2 CFD study

The default settings for implicit formulation and Roe-FDS flux type are kept. The explicit formulation is only normally used for cases where the characteristic time scale is of the same order as the acoustic time scale, for example the propagation of high Mach number shock waves, the implicit formulation instead is more stable and can be driven much harder to reach a converged solution in less time.

The gradient method for the spatial discretization is changed to Green-Gauss Node Based, being this slightly more computationally expensive than the other methods but more accurate. Second Order Upwind for flow and turbulence discretization are chosen, to accurately predict drag.

Solution controls

The default values for the Courant number (CFL) and under-relaxation factors (URFs) are kept. As we will be using automatic ‘solution steering’, the choice of CFL at this stage is not important for this case.

Solution Monitors

- The residuals are printed to the console.
- The convergence targets are turned off by setting the Convergence Criterion to ‘none’ and instead the number of iterations desired is set.
- The drag and lift monitors are enabled.
Solution initialization

The flow field is initialized based on the inlet boundary.

Solution steering

In the Run Calculation panel first the case must be checked, if there are no warnings the number of iterations to carry out can be set (in this case 1000 iterations) and the solution steering can be toggle on, making sure that the flow type is incompressible.

![Run Calculation panel](image)

Figure 8.28: Last step before the calculations.

Solution steering uses ‘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. It employs robust first order discretization in the early-stages of the main computation, then blends to the more accurate second order schemes as the solution stabilizes.

Now the case is ready to begin with the calculation, to do so it must be exported to a .cas file which will be later run in the cluster.
9 Results and comparison

Here the results from all the simulations will be presented. It must be said that the results shown here are the best obtained with the computational resources that were affordable. Of course, there is the possibility to improve them but relevant improvements in mesh quality demand more compute power when it comes to initialize the case or solve it.

9.1 Results summary

9.1.1 PVAST 327 results

The results are shown more profoundly and explicit in the Annex document, here only some of the graphical results are displayed.

![Figure 9.1: Pressure distribution in surface (0°)](image1)
![Figure 9.2: Pressure distribution in surface (30°)](image2)

![Figure 9.3: Pressure contour in the mid-plane (60°)](image3)
![Figure 9.4: Pressure contour in the mid-plane (90°)](image4)
9.1.2 PVAST 196 results

The results are shown more profoundly and explicit in the Annex document, here only some of the graphical results are displayed.

![Figure 9.5: Pressure contour in the mid-plane (30°)](image)

![Figure 9.6: Pressure contour in the mid-plane (60°)](image)

<table>
<thead>
<tr>
<th>Experimental data</th>
<th>Numerical results</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha = 0^\circ$; 180 km/h</td>
<td>$C_L: 0.010$, $C_D: 0.050$</td>
</tr>
<tr>
<td>$\alpha = 30^\circ$; 140 km/h</td>
<td>$C_L: 0.930$, $C_D: 0.520$</td>
</tr>
<tr>
<td>$\alpha = 60^\circ$; 140 km/h</td>
<td>$C_L: 0.580$, $C_D: 0.900$</td>
</tr>
<tr>
<td>$\alpha = 90^\circ$; 140 km/h</td>
<td>$C_L: 0.0$, $C_D: 1.100$</td>
</tr>
</tbody>
</table>

Table 9.2: Results of the PVAST 196, compared with the experimental data.

9.1.3 PVHT 2160 results

The results are shown more profoundly and explicit in the Annex document, here only some of the graphical results are displayed.
9.1 Results summary

Figure 9.7: Turbulence viscosity in a intersecting mid-plane (20°)

Figure 9.8: Eddy viscosity in the mid-surface (60°)

<table>
<thead>
<tr>
<th>Experimental data</th>
<th>Numerical results</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha = 0^\circ$; 180 km/h</td>
<td>$C_L$</td>
</tr>
<tr>
<td></td>
<td>0</td>
</tr>
<tr>
<td>$\alpha = 20^\circ$; 140 km/h</td>
<td>0.680</td>
</tr>
<tr>
<td>$\alpha = 40^\circ$; 140 km/h</td>
<td>0.840</td>
</tr>
<tr>
<td>$\alpha = 60^\circ$; 140 km/h</td>
<td>0.530</td>
</tr>
</tbody>
</table>

Table 9.3: Results of the PVHT 2160, compared with the experimental data.

Looking at the results and comparing them to the values obtained from the experimental data one can conclude that in general the results are logical or at least they adjust to what it is expected for an object of such shape. As said numerous times the aim was not that the numerical results and the experimental values were identical, actually, this would be almost impossible to achieve, because the data was obtained for a flat plate shape and also not for the actual dimensions, instead scale models were used in order to cover the large range of aspect ratios.

Taking into consideration that the geometry presented discontinuities and sharp edges, it was expected that the turbulence were high and thus the drag and lift coefficient were affected by that. It is seen, for example, in figure 9.9 that turbulence is generated by the presence of these edges.
9.2 Budget summary

An estimation was done in order to have a notion of the budget of this study, here not only the hours dedicated by the author of the study are taken into account, also concepts like the power consumption of the hardware used and the cost of this hardware and the software are considered. The assumptions made for the calculations are clearly stated in the Budget document.

<table>
<thead>
<tr>
<th>Concept</th>
<th>h</th>
<th>€/h</th>
<th>€</th>
</tr>
</thead>
<tbody>
<tr>
<td>Development of the project</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Information research</td>
<td>76</td>
<td>12</td>
<td>912</td>
</tr>
<tr>
<td>Setting of the cases</td>
<td>310</td>
<td>12</td>
<td>3720</td>
</tr>
<tr>
<td>Writing of the report</td>
<td>143</td>
<td>12</td>
<td>1716</td>
</tr>
<tr>
<td>Subtotal</td>
<td>529</td>
<td></td>
<td>6348</td>
</tr>
<tr>
<td>Hardware amortization</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Laptop</td>
<td>529</td>
<td>0.0132</td>
<td>7.01</td>
</tr>
<tr>
<td>Computational cluster</td>
<td>350</td>
<td>0.0137</td>
<td>4.79</td>
</tr>
<tr>
<td>Subtotal</td>
<td></td>
<td></td>
<td>11.80</td>
</tr>
<tr>
<td>Software amortization</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ANSYS FLUENT</td>
<td>350</td>
<td>0.6849</td>
<td>239.73</td>
</tr>
<tr>
<td>Energy consumption</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Laptop</td>
<td>529</td>
<td>0.0098</td>
<td>5.16</td>
</tr>
<tr>
<td>Computational cluster</td>
<td>350</td>
<td>0.09</td>
<td>31.5</td>
</tr>
<tr>
<td>Subtotal</td>
<td></td>
<td></td>
<td>36.66</td>
</tr>
<tr>
<td>Summary of the costs</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total cost</td>
<td></td>
<td></td>
<td>6636.18</td>
</tr>
<tr>
<td>Overhead</td>
<td></td>
<td></td>
<td>663.62</td>
</tr>
<tr>
<td>Final cost</td>
<td></td>
<td></td>
<td>7299.80</td>
</tr>
</tbody>
</table>

Table 9.4: Total budget of the project.
9.3 Economic feasibility of the study

How it has been already commented in the subsection 4.1, there are many reasons that makes one rely on CFD studies before carrying out the construction of a prototype, many of these are economical, helping to save money either reducing the costs of new designs or eliminating the need of controlled experiments which in some cases are very difficult or impossible to perform. Economically speaking and from the perspective of a company, although the price of the software used in this study can be seen as too high or not affordable for small companies, it is clear that it can only bring benefits to the company in design projects or optimization of products.

One must consider also the cost of suitable hardware, which is very important due to the fact that it can be the limiting factor of the accuracy of the study or the size of the case of study. Furthermore, the cost of the annual maintenance of this hardware have to be consider. Clearly the investment costs of a CFD capability are not small, but the total expense is not normally as great as that of a high quality experimental facility.

As an alternative there is non proprietary software like OpenFoam. However, though similar results as the ones from FLUENT or any other proprietary software can be achieved, it requires more expertise and qualified people to run the simulations and communicate their results. All in all, it is a tool which in the future with the development of computers will be of great help in the engineering field, more than it is now.

9.4 Planning of the project

9.4.1 Tasks identification from work breakdown structure (WBS)

A WBS diagram has been designed as shown in figure 9.10.
9.4.2 Brief tasks description

Table 9.5: Brief tasks description

<table>
<thead>
<tr>
<th>Code of task</th>
<th>Task identification</th>
<th>Brief description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Project Management</td>
<td></td>
</tr>
<tr>
<td>0.1</td>
<td>Planning</td>
<td></td>
</tr>
<tr>
<td>0.1.1</td>
<td>Work Breakdown Structure</td>
<td>Determine the WBS and the tasks to be done</td>
</tr>
<tr>
<td>0.1.2</td>
<td>Interdependences of the tasks</td>
<td>Determine the tasks interdependences and estimate their duration</td>
</tr>
<tr>
<td>0.2</td>
<td>Scheduling</td>
<td></td>
</tr>
<tr>
<td>0.2.1</td>
<td>Gantt diagram</td>
<td>Make the Gantt diagram in order to organize the work to accomplish the deadline</td>
</tr>
<tr>
<td>0.3</td>
<td>Quality Control</td>
<td></td>
</tr>
<tr>
<td>0.3.1</td>
<td>Supervision of the work</td>
<td>Ensure the correct development of the tasks</td>
</tr>
<tr>
<td>1</td>
<td>Previous concepts</td>
<td></td>
</tr>
<tr>
<td>1.1</td>
<td>Physics of the problem</td>
<td></td>
</tr>
<tr>
<td>1.1.1</td>
<td>Aerodynamic loads</td>
<td>Aerodynamic loads induced by the wind</td>
</tr>
<tr>
<td>1.1.2</td>
<td>Basic equations</td>
<td>Fundamental equations of fluid mechanics</td>
</tr>
<tr>
<td>1.2</td>
<td>Definition of the problem</td>
<td></td>
</tr>
<tr>
<td>1.2.1</td>
<td>Dimensions</td>
<td>Dimensions of the solar tracker</td>
</tr>
<tr>
<td>1.2.2</td>
<td>Wind conditions</td>
<td>Estimation of the wind conditions</td>
</tr>
<tr>
<td>1.2.3</td>
<td>Cases of study</td>
<td>Diverse cases of study that will be considered in the study</td>
</tr>
<tr>
<td>2</td>
<td>Analytical Approach</td>
<td></td>
</tr>
<tr>
<td>2.1</td>
<td>Simplification of the problem</td>
<td></td>
</tr>
<tr>
<td>2.1.1</td>
<td>Assumptions and simplifications</td>
<td>Assumptions considered in order to simplify the model</td>
</tr>
<tr>
<td>2.1.2</td>
<td>Aerodynamic data</td>
<td>Selection of aerodynamic data according to the assumptions already stated</td>
</tr>
<tr>
<td>2.1.3</td>
<td>Wind profile</td>
<td>Wind profile that will be used in the analytical resolution</td>
</tr>
<tr>
<td>2.2</td>
<td>Results exposition</td>
<td></td>
</tr>
<tr>
<td>2.2.1</td>
<td>Compilation of the results for different wind velocities</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>CFD Approach</td>
<td></td>
</tr>
<tr>
<td>3.1</td>
<td>Fundamental concepts</td>
<td>Fundamental concepts that has to be taken into account before undertaking a CFD study</td>
</tr>
<tr>
<td>3.1.1</td>
<td>Turbulence</td>
<td>Explanation of the turbulence concept</td>
</tr>
</tbody>
</table>

(continues on next page)
### 9.4 Planning of the project

#### Table 9.5: Brief tasks description (continued from previous page)

<table>
<thead>
<tr>
<th>Code of task</th>
<th>Task identification</th>
<th>Brief description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1.2</td>
<td>Turbulence models</td>
<td>Review of the diverse turbulence models in CFD</td>
</tr>
<tr>
<td>3.1.3</td>
<td>Boundary layer</td>
<td>Explanation of the boundary layer concept</td>
</tr>
<tr>
<td>3.1.4</td>
<td>Meshing</td>
<td>Explanation of the meshing concept</td>
</tr>
<tr>
<td>3.1.5</td>
<td>Boundary conditions</td>
<td>List of boundary conditions eventually stating the ones used present in the case of study</td>
</tr>
<tr>
<td>3.2</td>
<td>CFD study</td>
<td></td>
</tr>
<tr>
<td>3.2.1</td>
<td>3-D Modelling</td>
<td>Creation of the geometry</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Mesh definition</td>
<td>Justification of the mesh used</td>
</tr>
<tr>
<td>3.2.3</td>
<td>Meshing process</td>
<td>Creation of the mesh</td>
</tr>
<tr>
<td>3.2.4</td>
<td>Boundary conditions setting</td>
<td>Boundary conditions that are set for the numerical simulation and its justification</td>
</tr>
<tr>
<td>3.2.5</td>
<td>Case set-up</td>
<td>Procedure followed for the set-up of the cases.</td>
</tr>
<tr>
<td>3.3</td>
<td>Results and comparison</td>
<td></td>
</tr>
<tr>
<td>3.3.1</td>
<td>Compilation of results</td>
<td>Clear presentation of the results</td>
</tr>
<tr>
<td>3.3.2</td>
<td>Validation</td>
<td>Validation of the results, comparing them with some reference</td>
</tr>
<tr>
<td>3.3.3</td>
<td>Analysis and discussion</td>
<td>Analysis and discussion of the results</td>
</tr>
<tr>
<td>3.4</td>
<td>Conclusions</td>
<td></td>
</tr>
<tr>
<td>3.4.1</td>
<td>Initial conclusions</td>
<td>Conclusions about the results and the experience of the project</td>
</tr>
<tr>
<td>3.4.2</td>
<td>Continuity recommendations</td>
<td>Comments about modifications proposed and how the study can be continued</td>
</tr>
</tbody>
</table>

#### 9.4.3 Interdependency relationship among tasks

#### Table 9.6: Interdependency relationship among tasks

<table>
<thead>
<tr>
<th>Code of task</th>
<th>Task identification</th>
<th>Preceding task(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1.1</td>
<td>Work Breakdown Structure</td>
<td>None</td>
</tr>
<tr>
<td>0.1.2</td>
<td>Interdependences of the tasks</td>
<td>0.1.1</td>
</tr>
<tr>
<td>0.2.1</td>
<td>Gantt diagram</td>
<td>0.1.2</td>
</tr>
<tr>
<td>0.4.1</td>
<td>Supervision of the work</td>
<td>0.2.1</td>
</tr>
<tr>
<td>1.1.1</td>
<td>Aerodynamic loads</td>
<td>0.2.1</td>
</tr>
<tr>
<td>1.1.2</td>
<td>Basic equations</td>
<td>1.1.1</td>
</tr>
<tr>
<td>1.2.1</td>
<td>Dimensions</td>
<td>1.2.2</td>
</tr>
<tr>
<td>1.2.2</td>
<td>Wind conditions</td>
<td>1.2.1</td>
</tr>
<tr>
<td>1.2.3</td>
<td>Cases of study</td>
<td>1.2.2</td>
</tr>
<tr>
<td>2.1.1</td>
<td>Assumptions and simplifications</td>
<td>1.2.3</td>
</tr>
<tr>
<td>2.1.2</td>
<td>Aerodynamic data</td>
<td>2.1.1</td>
</tr>
<tr>
<td>2.1.3</td>
<td>Wind profile</td>
<td>2.1.2</td>
</tr>
<tr>
<td>2.2.1</td>
<td>Results exposition</td>
<td>2.1.3</td>
</tr>
<tr>
<td>3.1.1</td>
<td>Turbulence</td>
<td>2.2.1</td>
</tr>
<tr>
<td>3.1.2</td>
<td>Turbulence models</td>
<td>3.1.1</td>
</tr>
<tr>
<td>3.1.3</td>
<td>Boundary layer</td>
<td>3.1.2</td>
</tr>
<tr>
<td>3.1.4</td>
<td>Meshing</td>
<td>3.1.3</td>
</tr>
<tr>
<td>3.1.5</td>
<td>Boundary conditions</td>
<td>3.1.4</td>
</tr>
<tr>
<td>3.2.1</td>
<td>3-D Modelling</td>
<td>3.1.5</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Mesh definition</td>
<td>3.2.1</td>
</tr>
<tr>
<td>3.2.3</td>
<td>Meshing process</td>
<td>3.2.2</td>
</tr>
<tr>
<td>3.2.4</td>
<td>Boundary conditions setting</td>
<td>3.2.5</td>
</tr>
<tr>
<td>3.2.5</td>
<td>Case set-up</td>
<td>3.4.1</td>
</tr>
<tr>
<td>3.3.1</td>
<td>Compilation of results</td>
<td>3.2.5</td>
</tr>
<tr>
<td>3.3.2</td>
<td>Validation</td>
<td>3.3.1</td>
</tr>
<tr>
<td>3.3.3</td>
<td>Analysis and discussion</td>
<td>3.3.2</td>
</tr>
<tr>
<td>3.4.1</td>
<td>Initial conclusions</td>
<td>3.3.3</td>
</tr>
<tr>
<td>3.4.2</td>
<td>Continuity recommendations</td>
<td>3.4.1</td>
</tr>
</tbody>
</table>
9.4.4 Level of effort to develop each task

Table 9.7: level of effort

<table>
<thead>
<tr>
<th>Code of task</th>
<th>Gantt code</th>
<th>Task name</th>
<th>Level of effort</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1.1</td>
<td>1</td>
<td>Work Breakdown Structure</td>
<td>1 day</td>
</tr>
<tr>
<td>0.1.2</td>
<td>2</td>
<td>Interdependences of the tasks</td>
<td>1 day</td>
</tr>
<tr>
<td>0.2.1</td>
<td>3</td>
<td>Gantt diagram</td>
<td>2 days</td>
</tr>
<tr>
<td>0.3.1</td>
<td>28</td>
<td>Supervision of the work</td>
<td>25 days</td>
</tr>
<tr>
<td>1.1.1</td>
<td>4</td>
<td>Aerodynamic loads</td>
<td>2 days</td>
</tr>
<tr>
<td>1.1.2</td>
<td>5</td>
<td>Basic equations</td>
<td>1 day</td>
</tr>
<tr>
<td>1.2.1</td>
<td>6</td>
<td>Dimensions</td>
<td>2 days</td>
</tr>
<tr>
<td>1.2.2</td>
<td>7</td>
<td>Wind conditions</td>
<td>2 days</td>
</tr>
<tr>
<td>1.2.3</td>
<td>8</td>
<td>Cases of study</td>
<td>3 days</td>
</tr>
<tr>
<td>2.1.1</td>
<td>9</td>
<td>Assumptions and simplifications</td>
<td>5 days</td>
</tr>
<tr>
<td>2.1.2</td>
<td>10</td>
<td>Aerodynamic data</td>
<td>10 days</td>
</tr>
<tr>
<td>2.1.3</td>
<td>11</td>
<td>Wind profile</td>
<td>1 day</td>
</tr>
<tr>
<td>2.1.4</td>
<td>12</td>
<td>Analytical results</td>
<td>1 day</td>
</tr>
<tr>
<td>3.1.1</td>
<td>13</td>
<td>Turbulence</td>
<td>1 day</td>
</tr>
<tr>
<td>3.1.2</td>
<td>14</td>
<td>Turbulence models</td>
<td>2 days</td>
</tr>
<tr>
<td>3.1.3</td>
<td>15</td>
<td>Boundary layer</td>
<td>1 day</td>
</tr>
<tr>
<td>3.1.4</td>
<td>16</td>
<td>Meshing</td>
<td>1 day</td>
</tr>
<tr>
<td>3.1.5</td>
<td>17</td>
<td>Boundary conditions</td>
<td>1 day</td>
</tr>
<tr>
<td>3.2.1</td>
<td>18</td>
<td>3-D Modelling</td>
<td>1 day</td>
</tr>
<tr>
<td>3.2.2</td>
<td>19</td>
<td>Mesh definition</td>
<td>5 days</td>
</tr>
<tr>
<td>3.2.3</td>
<td>20</td>
<td>Meshing process</td>
<td>19 days</td>
</tr>
<tr>
<td>3.2.4</td>
<td>21</td>
<td>Boundary conditions setting</td>
<td>3 days</td>
</tr>
<tr>
<td>3.2.5</td>
<td>22</td>
<td>Case set-up</td>
<td>2 days</td>
</tr>
<tr>
<td>3.3.1</td>
<td>23</td>
<td>Compilation of results</td>
<td>1 day</td>
</tr>
<tr>
<td>3.3.2</td>
<td>24</td>
<td>Validation</td>
<td>3 days</td>
</tr>
<tr>
<td>3.3.3</td>
<td>25</td>
<td>Analysis and discussion</td>
<td>3 days</td>
</tr>
<tr>
<td>3.4.1</td>
<td>26</td>
<td>Initial conclusions</td>
<td>2 days</td>
</tr>
<tr>
<td>3.4.2</td>
<td>27</td>
<td>Continuity recommendations</td>
<td>2 days</td>
</tr>
</tbody>
</table>

Note: The level of effort is counted in terms of days of dedication to that task
9.5 Conclusions and continuity recommendations

All in all, it can be said that the main goals of the project were achieved within the expected time-line.

In the first stage of the project, an extensive investigation of the key aspects of the simulation of fluid dynamics was performed. The creation of a simple and comprehensive knowledge database was the goal of this part of the project, this would allow the inexperienced user to understand some of the backgrounds of the simulation as well as give him some advice for the proper modelling of his problem. An special focus was given on the turbulence treatment as it is still one of the main difficulties that CFD calculations must deal with.

The second part consisted in the use of the software packages ANSYS, in order to simulate the flow around the solar panels. It has been very satisfying that all of the aspects investigated during the first part of the project could successfully be applied in a careful modelling of the geometry and flow conditions around these elements. This represents also that the author has acquired important knowledge about CFD studies and that it is given by the experience itself as a CFD user.
The results show good agreement to the studied values, the aerodynamic coefficients extracted from the experimental study (REF). It could be proven that the simulation could reliably predict the uplift forces (which are the most important parameter for this type of elements) within a very reasonable amount of time compared to a classical wind tunnel investigation.

Regarding what could be done in order to extent this study there are some proposals, following the aerodynamic study and once the induced aerodynamic forces for every case are clear, it is logical to proceed with structural analysis in order to ensure the structural stability of the whole supporting framework. From here on, modifications in the structure can be thought and tested or it can be even completely re-designed, dimensioning the fundamental structural elements making use of the results obtained here.

This would be a new study relying on previous work focusing on the structural aspect, nonetheless this study can be extended in aerodynamic aspects also, considering concepts which have not been treated in this study, for example the interaction between between solar trackers when they are displayed as an array, or what is the effect of fences, or elements which could have some kind of sheltering effect, in the aerodynamic forces induced in the solar tracker. This study will focus more in ways that could reduce or have an important effect on the magnitude of aerodynamic features of the photovoltaic trackers.

In fact some of the ideas just mentioned, were having in mind to be implemented but due to the lack of time and the poor experience that the author had with the ANSYS software they could not be applied, meaning that a considerable amount of time was dedicated to the understanding and learning of the software, specially the meshing software ICEM CFD, this fact is portrayed very clearly in the table 9.7 where it can be seen that the dedication to the meshing process is higher than for the other tasks, this was not planned before beginning with the project and thus additional or optional tasks were thought to be done, the main reason behind this bad planning was the lack of knowledge of the implications of a CFD study.

In order to conclude, it must be said that CFD calculations show an increasing potential for application in many fields of wind engineering as it has been shown in this study. Nevertheless, before carrying out any study of these characteristics, a careful study of the backgrounds of fluid mechanics as well as CFD modelling is strongly advised in order to make the right choices regarding turbulence, meshing and solver type for a given problem.
Section 10: References

10 References


