Experimental and CFD Analysis of a Biplane Wells Turbine for Wave Energy Harnessing

Master Thesis

João Sousa Alves
Abstract

Several alternative ways of producing energy came up as the world took conscience of the finite availability of fossil fuels and the environmental consequences of its use and processing. Wave and tidal energy are among the so called green energies. Wave energy converters have been under research for the past two decades and yet there hasn’t been one technology that gathered everyone’s acceptance as being the most suitable one.

The present work is focused on a self-rectifying turbine for wave energy harnessing. It’s a self-rectifying biplane Wells with an intermediate stator. The main goal is to evaluate the performance of such a turbine. Two different analyses were performed: experimental and computational.

The experimental tests were made so that efficiency, velocity profiles and loss coefficients could be calculated. To do so scaled-down prototypes were built from scratch and tested experimentally.

The 3D numerical analysis was possible by using a CFD commercial code: Fluent 6.3. Several simulations were performed for different flow coefficients. Three different degrees of mesh refinement were applied and k-ε turbulence model was the one chosen to simulate the viscous behavior of the flow through the turbine. A steady-state analysis is due and two mixing planes were used at the interfaces between the rotors and the stator.

In the end comparisons are made between the experimental and numerical results.
Acknowledgments

I’d like to express my sincere gratefulness to: my scientific coordinator, Professor. Luís Gato, for all his help, availability in answering all the questions and for the continuous improvement feedback that much contributed for the enrichment of the present work.

The experimental and numerical part of this work was performed at Instituto Superior Técnico more specifically at Engineering Mechanics department. I would like to thank this institution for providing all the means necessary for the completion of this thesis.

To Eng. Rui Gomes for his remarkable and unconditional help that was crucial on both experimental and computational part of this work.

To Dr. João Henriques for all the technical and theoretical advices throughout this work.

To everyone working at 1st floor of “Pavilhão de Mecânica IV” for providing a good working environment.

Last but not least to everyone at the Thermo-fluid workshop for all the advices and efficient work regarding the manufacturing and assembly of the experimental rig.

To Professor Damian Vogt for accepting being my KTH examiner and also for all the suggestions made during the execution of this thesis.
Index

Acknowledgments ........................................................................................................ vi
List of figures .............................................................................................................. x
List of abbreviations .................................................................................................. xii
List of symbols .......................................................................................................... xiii
List of subscripts ....................................................................................................... xiv

1 Introduction ............................................................................................................. 1
  1.1 Overview ............................................................................................................ 1
  1.2 World Energetic Context and the Wave Energy Alternative .................................. 1
  1.3 Wave Energy Converters - Oscillating Water Column ........................................ 2
  1.4 Self-Rectifying Turbines for OWC Devices .......................................................... 3
  1.5 Present Contribution .......................................................................................... 7

2 Turbines Description .............................................................................................. 8
  2.1 General Description .......................................................................................... 8
  2.2 Manufacturing process ...................................................................................... 9
  2.3 The rotor ........................................................................................................... 10
    2.3.1 Rotor blades and Wells Turbine .................................................................. 10
    2.3.2 Rotors for “4_19_4” turbine model and “no guide vanes” configurations ...... 12
  2.4 The guide vanes ................................................................................................ 12
  2.5 Working and thermodynamic basic principles .................................................... 13

3 Experimental Analysis ........................................................................................... 17
  3.1 Experimental rig ............................................................................................... 17
  3.2 Instrumentation .................................................................................................. 18
  3.3 Data acquisition ................................................................................................. 20
    3.3.1 Efficiency, pressure drop and torque coefficients ....................................... 20
    3.3.2 Probe Calibration ....................................................................................... 22
    3.3.3 Flow angles ............................................................................................... 23

4 Experimental Results ............................................................................................ 25
  4.1 Transverse probe calibration ............................................................................. 25
List of tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 3.1</td>
<td>Pressure manometers scales</td>
<td>18</td>
</tr>
<tr>
<td>Table 4.1</td>
<td>Calibration coefficients trend line equations</td>
<td>26</td>
</tr>
<tr>
<td>Table 4.2</td>
<td>Friction losses trend line equations</td>
<td>27</td>
</tr>
<tr>
<td>Table 4.3</td>
<td>Manufacturer’s bias specifications</td>
<td>31</td>
</tr>
<tr>
<td>Table 4.4</td>
<td>Uncertainty results for the “4_19_4” configuration</td>
<td>32</td>
</tr>
<tr>
<td>Table 4.5</td>
<td>Uncertainty results for the “no guide vanes” configuration</td>
<td>32</td>
</tr>
<tr>
<td>Table 4.6</td>
<td>Loss coefficients over different stages of the turbine for three different flow coefficients</td>
<td>39</td>
</tr>
<tr>
<td>Table 4.7</td>
<td>Comparison between data read from efficiency test and calculated data using integration schemes for the “4_19_4” turbine</td>
<td>40</td>
</tr>
<tr>
<td>Table 4.8</td>
<td>Loss coefficients over different stages of the turbine for three different flow coefficients</td>
<td>40</td>
</tr>
<tr>
<td>Table 4.9</td>
<td>Comparison between data read from efficiency test and calculated data using integration schemes for the “no guide vanes” turbine</td>
<td>41</td>
</tr>
<tr>
<td>Table 5.1</td>
<td>Equation discretization types</td>
<td>46</td>
</tr>
<tr>
<td>Table 5.2</td>
<td>Element distribution and total number of elements for the 4_19_4 turbine model</td>
<td>49</td>
</tr>
<tr>
<td>Table 5.3</td>
<td>Element skewness ratio in terms of total number of elements for the 4_19_4 turbine</td>
<td>50</td>
</tr>
<tr>
<td>Table 5.4</td>
<td>Element distribution for the “no guide vanes” configuration</td>
<td>50</td>
</tr>
<tr>
<td>Table 5.5</td>
<td>Element skewness ratio in terms of total number of elements for the “no guide vanes” configuration</td>
<td>51</td>
</tr>
<tr>
<td>Table 5.6</td>
<td>Mixing plane boundary conditions</td>
<td>53</td>
</tr>
<tr>
<td>Table 6.1</td>
<td>Properties integration differences across mixing-planes for the structured mesh G2S</td>
<td>58</td>
</tr>
<tr>
<td>Table 6.2</td>
<td>Tip gap effect on peak efficiency</td>
<td>63</td>
</tr>
<tr>
<td>Table 6.3</td>
<td>Parameter variance with mesh refinement</td>
<td>63</td>
</tr>
<tr>
<td>Table 6.4</td>
<td>Parameter uncertainty for finest mesh G1S</td>
<td>63</td>
</tr>
<tr>
<td>Table 6.5</td>
<td>Parameter variance with mesh refinement</td>
<td>63</td>
</tr>
<tr>
<td>Table 6.6</td>
<td>Parameter uncertainty for finest mesh</td>
<td>64</td>
</tr>
</tbody>
</table>

Appendix Table 1- Stator data ........................................................................................................ X

List of figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1</td>
<td>A) Wells turbine with guide vanes; B) Wells turbine with variable pitch</td>
<td>4</td>
</tr>
<tr>
<td>Figure 1.2</td>
<td>A) Biplane Wells turbine with guide vanes; B) Contra-rotating biplane Wells turbine</td>
<td>4</td>
</tr>
<tr>
<td>Figure 1.3</td>
<td>A) Monoplane Wells turbine; B) Biplane Wells turbine</td>
<td>5</td>
</tr>
<tr>
<td>Figure 1.4</td>
<td>A) Impulse turbine with variable pitch guide vanes; B) Impulse turbine with fixed guide vanes</td>
<td>6</td>
</tr>
</tbody>
</table>
Figure 1.5- McCormick turbine ................................................................. 6
Figure 2.1- "4_19_4" turbine model main dimensions and section overview. The stator blade shown does not reproduce the real geometry, being only representative. (All dimensions are in mm) ........................................... 8
Figure 2.2- Turbine models in a x- @ plane .................................................. 9
Figure 2.3- Wells turbine velocity triangle and forces diagram ........................................ 11
Figure 2.4- "4_19_4" turbine velocity triangles .................................................................. 14
Figure 2.5- "No guide vanes" turbine velocity triangles .................................................. 15
Figure 3.1- Experimental rig ............................................................................. 17
Figure 3.2- Turbine model ................................................................................. 18
Figure 3.3- Transverse probe assembled on turbine model ............................................... 19
Figure 3.4-a) Transverse probe front view; b) Transverse probe lateral view ...................... 20
Figure 4.1- Pressures read by the probe relative to total pressure versus angle ................... 25
Figure 4.2- Calibration coefficients versus angle ....................................................... 26
Figure 4.3- Friction torque versus angular velocity ................................................... 26
Figure 4.4- Efficiency curves for several angular velocities ........................................... 27
Figure 4.5- Total pressure drop coefficient for several angular velocities ......................... 28
Figure 4.6- Torque coefficient for several angular velocities ........................................ 28
Figure 4.7- Efficiency curves for several angular velocities ......................................... 29
Figure 4.8- Total pressure drop coefficient for several angular velocities ......................... 29
Figure 4.9- Torque coefficient for several angular velocities ........................................ 30
Figure 4.10- Torque signal sample for a time period of 2 seconds ................................... 33
Figure 4.11- Power spectral density ........................................................................... 33
Figure 4.12- Torque signal after applying the low pass filter ........................................... 34
Figure 4.13- Absolute flow angle downstream the first rotor for several flow coefficients .... 35
Figure 4.14- Absolute flow angle downstream the stator for several flow coefficients ...... 36
Figure 4.15- Absolute flow angle downstream the second rotor for several flow coefficients 36
Figure 4.16- Absolute flow angle between rotors for several flow coefficients .................. 37
Figure 4.17- Absolute flow angle downstream the second rotor for several flow coefficients 37
Figure 5.1- "4_19_4" computational domain (casing and periodic surfaces not represented) .... 47
Figure 5.2-a) C-type mesh around the rotor blade, b) Rotor blade mesh ....................... 48
Figure 5.3- Stator hub and blade mesh ........................................................................ 49
Figure 5.4- Blade tip mesh detail on leading and trailing edges ...................................... 51
Figure 5.5- Mixing plane locations ............................................................................ 53
Figure 5.6- Periodic boundaries location (all blue lateral surfaces) ................................ 54
Figure 5.7- Turbine with no stator domain .................................................................. 55
Figure 6.1 - Y+ values for one flow coefficient of each configuration. "No guide vanes" on the left hand-side and "4_19_4" on the right hand-side ........................................... 57
Figure 6.2- Comparison between experimental efficiency curves and numerical efficiency curve ............................................................... 59
FIGURE 6.3- TOTAL PRESSURE DROP COEFFICIENT COMPARISON BETWEEN EXPERIMENTAL RESULTS AND NUMERICAL RESULTS ........... 59
FIGURE 6.4- TORQUE COEFFICIENT COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS........................................... 60
FIGURE 6.5- EFFICIENCY CURVE COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS ........................................... 60
FIGURE 6.6- TOTAL PRESSURE DROP COEFFICIENT COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS .......................... 61
FIGURE 6.7- TORQUE COEFFICIENT COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS ........................................... 61
FIGURE 6.8- ABSOLUTE FLOW ANGLES DOWNSTREAM THE FIRST ROTOR. COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS .......................................................................................................................................................... 62
FIGURE 6.9- ABSOLUTE FLOW ANGLES DOWNSTREAM THE SECOND ROTOR. COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS .......................................................................................................................................................... 62

APPENDIX FIGURE 1- EFFICIENCY AND TOTAL PRESSURE DROP COEFFICIENT CURVES (FILLED SQUARES AND TRIANGLES) ............... I
APPENDIX FIGURE 2- TORQUE COEFFICIENT CURVE (FILLED TRIANGLES) ........................................................................................................... I
APPENDIX FIGURE 3- HUB STEEL PLATE WITH GUIDING VANE CUTS BEFORE ROLL BENDING (PART 1/2)............................................................ II
APPENDIX FIGURE 4- HUB STEEL PLATE WITH GUIDING VANE CUTS BEFORE ROLL BENDING (PART 2/2).......................................................... III
APPENDIX FIGURE 5- CASING STEEL PLATE WITH GUIDING VANE CUTS BEFORE ROLL BENDING (PART 1/2)................................................... III
APPENDIX FIGURE 6- CASING STEEL PLATE WITH GUIDING VANE CUTS BEFORE ROLL BENDING (PART 2/2).................................................. IV
APPENDIX FIGURE 7- PLAN STATOR BLADE BEFORE ROLL BENDING........................................................................................................ IV
APPENDIX FIGURE 8- STEEL STRUCTURE THAT BEARS THE BEARINGS AND SHAFT. .................................................................................. V
APPENDIX FIGURE 9- SHAFT .............................................................................................................................................................................. V
APPENDIX FIGURE 10- STEEL BUSHING MAKING THE UNION BETWEEN STATOR AND STEEL STRUCTURE THAT HOLDS THE SHAFT ........ VI
APPENDIX FIGURE 11- ROTOR END PLATE ....................................................................................................................................................... VI
APPENDIX FIGURE 12- BEARING STOPPER................................................................................................................................................... VII
APPENDIX FIGURE 13- PART CONNECTING SHAFT AND ROTOR HUB........................................................................................................ VII
APPENDIX FIGURE 14- ROTOR HUB .............................................................................................................................................................. VIII
APPENDIX FIGURE 15- ROTOR END PLATE............................................................................................................................................. VIII
APPENDIX FIGURE 16- STEEL STRUCTURE THAT HOLDS THE TURBINE MODEL......................................................................................... IX

List of abbreviations

CFD: Computational Fluid Dynamics
CO2: Carbon Dioxide
DC: Direct Current
IST: Instituto Superior Técnico
NACA: National Advisory Committee for Aeronautics
OAPEC: Organization of Arab Petroleum Exporting Countries
OWC: Oscillating Water Column
rpm: Rotations Per Minute
PIV: Particle Image Velocimetry
PT: Pressure Tap
WEC: Wave Energy Converter
“4_19_4”: Turbine with 4 blades in each rotor and an intermediate stator with 19 blades
“no guide vanes”: Turbine with 4 blades in each rotor and no intermediate stator.

List of symbols

A  Area
A_n  Cross sectional area of the convergent nozzle
b  Bias error
C  Absolute velocity
c, C  Total profile chord, probe coefficient, turbulence constant
D  Drag
D_n  Hydraulic diameter
E  Energy
F  Force
G  “4_19_4” turbine configuration mesh
G_k  Turbulent kinetic energy
h  Enthalpy
H  Turbine with guide vanes configuration mesh
k  Nozzle flow loss coefficient
L  Lift
ṁ  Mass flow rate
p  Precision error, pressure
P_{tot}  Total pressure
Q  Volumetric flow rate
R, r  Radius, Reynolds tensor
R_1  Rotor 1 or upstream rotor
R_2  Rotor 2 or downstream rotor
s  Laplace transform variable
τ; T  Chord, Torque
T^*  Torque coefficient
u; U  x-velocity, uncertainty; rotational velocity
v, V  y-velocity; Volt, Velocity
\( w, W \quad z\text{-}velocity; \text{Relative velocity} \\
\( x \quad \text{Horizontal coordinate, x-coordinate} \\
\( y \quad \text{Vertical coordinate} \\
\( Y^+ \quad \text{Dimensionless wall distance} \\
\( \alpha \quad \text{Absolute velocity flow angle} \\
\( \beta \quad \text{Relative velocity flow angle} \\
\( \sigma \quad \text{Standard deviation} \\
\( \Delta p_0^\ast \quad \text{Total pressure drop coefficient} \\
\( \Omega, \omega \quad \text{Angular velocity} \\
\( \varepsilon \quad \text{Energy dissipation rate} \\
\( \zeta \quad \text{Loss coefficient} \\
\( \mu \quad \text{Air kinematic viscosity} \\
\( \rho \quad \text{Air density} \\
\( \phi \quad \text{Flow coefficient, flow property} \\

\textbf{List of subscripts} \\

0 \quad \text{Total condition, far upstream} \\
1 \quad \text{Downstream first rotor} \\
2 \quad \text{Downstream stator} \\
3 \quad \text{Downstream second rotor} \\
4 \quad \text{Far downstream} \\
7 \quad \text{Transverse probe lateral pressure tap} \\
8 \quad \text{Transverse central tap} \\
9 \quad \text{Transverse probe lateral pressure tap} \\
\text{dyn} \quad \text{Dynamic condition} \\
\text{t} \quad \text{Turbulent} \\
\text{tot} \quad \text{Total condition}
1 Introduction

1.1 Overview
The present work consists of a preliminary study of an alternative biplane Wells turbine configuration to be used in an Oscillating Water Column device for wave energy harnessing. This turbine differs from others in the way its rotors and stator are placed relatively to each other, i.e. the stator is located between the rotors. Three different configurations were manufactured:

1. Four blades rotors with a nineteen blades stator
2. Six blades rotors with a twenty-nine blades stator
3. Four blades rotors with guide vanes

However only two of the above three configurations were experimentally tested: number 1 and 3.

The experimental analysis was performed at Instituto Superior Técnico, Lisbon, Portugal. The facilities were supplied by the Engineering Mechanics Department – Thermo-fluids sub-division.

A computational fluid dynamics analysis was also performed to the experimentally tested turbines so that results could be compared.

In Chapter 1 one can find a brief description of the actual energetic situation and the space that wave energy fills in this context. A brief explanation about the OWC working principle is also addressed. Other types of Wells and impulse turbines and their characteristics are technically presented and finally the contribution of the present work to this field knowledge is explained.

Chapter 2 presents us a complete description of the tested turbines where its conception, geometry and velocity triangles throughout its stages are described.

Chapter 3 describes the experimental rig and its different components as well as the data acquisition equipment and procedure.

Chapter 4 is where one can find the experimental results namely efficiency curves, pressure and torque coefficient curves, velocity triangles and other transverse probe calibration results.

Chapter 5 addresses the main parameters of the computational fluid dynamics analysis: control volume and mesh generation, calculation method and boundary conditions setting. Different meshes are described and summarized.

In Chapter 6 main numerical results are presented and discussed. Integral parameters as well as their error analysis are also taken into account.

Chapter 7 is where conclusions are taken, both analyzing the experimental results between configurations 1 and 3 and their CFD numerical model.

1.2 World Energetic Context and the Wave Energy Alternative

Social awareness regarding energetic issues has been gradually arising among nowadays societies. It is now broadly discussed the way we produce energy and the consequences its price, access and
distribution may have on everyone’s way of life. The first time world realized the impact of an unstable oil supply was in 1973. The oil embargo created by OPEC (Organization of Arab Petroleum Exporting Countries) lasted for a few months. During that period the oil prices ascended dramatically and considerable economic and social drawbacks were expected. As a consequence of this historical fact one realized the high fossil fuel dependence developed countries seemed to have. This acted as launching ramp for alternative ways to produce energy. However the main concern today regarding this same topic is not only the possibility of a limited natural availability of fossil fuels but also the consequences of its use on the environment. It has been estimated that by 2007 about 86% of world total energy needs were supplied by oil, coal and gas [1]. All the latter sources of energy imprint heavy environmental marks during its processing and consumption. Moreover such a supply can’t be delivered at the rate that these fuels are being consumed. Therefore renewable energies rose as a promising way to keep on supplying world’s energy demand. The investment has been clearly intensified over the past two decades and political will to change the actual situation is also noticeable. As an example of such one can look at the fact that European Union set the year 2020 as target to have reached a share of 20% of total electrical energy produced coming from renewable sources as well as a 20% cut off in CO2 emissions [2]. This goal will be achieved mainly based on solar, wind, hydroelectric and geothermal sources. However wave and tidal energy have been under a lot of researching efforts and has also been emerging among the so-called green energies. During an off-shore located storm winds can perturb the sea surface. The wind momentum is transferred to the water as long as its waves move slower than the wind passing by. These phenomena create crests that evolve into swells. These groups of waves can travel in deep-waters with minor energy losses. Waves transport both kinetic and geo-potential energy proportional to the wind intensity, period of time acting on a portion of water and the distance over which this interaction takes place. It is estimated that the total amount of energy carried by waves is about 2 TW[3]. This resource is not evenly distributed among the globe. Measurements over a period of 10 years have been performed and some information could be analyzed. The hotspots seem to be located between 40 and 60° of latitude on both north and south hemispheres [4]. Also there is always a variance that depends on the season and period of the year considered. It becomes obvious that it’s not possible to harness all this energetic potential. Technical and deploying place issues act as important limitations. However even a small portion of this value can make an important contribution to the total energy mix.

1.3 Wave Energy Converters- Oscillating Water Column

There is a considerable number of different Wave Energy Converters for possible use or still under research¹. Unlike what happened with the wind energy industry with the three-bladed turbines, one WEC configuration hasn’t gathered together everyone’s acceptance.

¹ See [15] for a review on different WEC under research nowadays.
Most of OWC installations are today located on the shoreline. This type of devices was one of the first ones to be seriously considered for wave energy conversion. The principle is based on a semi-submerged air chamber in which a current of air is forced back and forth by the incident waves. This air current occurs because the chamber has two open ends: one in contact with water and the other one in contact with the atmosphere. When a wave hits the structure the chamber which has both air and water blows away the air as water level rises. In between two wave crests the water level falls and air is sucked in again to the chamber. An air turbine, through which the flow passes by, drives an electric generator through the shaft. Therefore rotational mechanical energy is transformed into electrical energy available for the grid. It is important to shape the air chamber in order to closely get resonance conditions and therefore better use the available energy resource. In order to avoid an over pressurized air chamber that would compromise an efficient and turbine operation a relief valve is mounted and acts whenever the sea conditions require so. This valve also ensures the mechanical integrity of the turbine in case of extreme incident sea conditions.

This has been one of the most intensively studied WEC apparatus; countries like the UK, Spain, India, China, Japan, Portugal and Norway are good examples of such effort [4] [5]. Two examples of full-scale models were set at Pico Island (Archipelago of The Azores, Portugal) and Islay, Scotland. Even though the turbine structures for energy production are different the general working principle is essentially the same.

OWC devices are also possible of being set offshore. It has been estimated that an on shoreline WEC only captures 25 to 50% of the resource comparing to an offshore located one [6]. However such a configuration has all the disadvantages of being far from the shoreline. It becomes hard to first install it as well as to run maintenance operations from then on. Also very long underwater electrical cables would be necessary to drive electricity to an onshore power station. Nevertheless this hypothesis is far from disregarded because of such motives.

### 1.4 Self-Rectifying Turbines for OWC Devices

Several turbine configurations have been addressed when it comes to its structure. Different types and relative locations of rotor and stator may apply. Impulse turbine and Wells turbine have however gathered most of the efforts in order to find a consensual solution. Both these turbines can be self-rectifying in the sense that the rotor is always rotating in the same direction no matter the direction of the incident approaching flow. The advantage of this feature becomes evident in OWC applications because of the oscillatory pattern of the flow. However the way energy is transferred from flowing air to the rotor differ in these two applications. The Wells turbine is based on the pressure drop whereas an impulse turbine is designed to extract kinetic energy from the flow. The latter also means that in design conditions no expansion exists through an impulse turbine rotor and therefore the static pressures up and downstream of the rotor are barely the same.

According to [7] and [8] one next presents some more details on different setups for these two types of turbines. As one can observe (Figures 1.1 and 1.2) there can be different setups for the Wells turbine. Specifically we can look at the following six:
- Wells turbine with guide vanes
- Wells turbine with variable pitch rotor blades
- Biplane Wells turbine with guide vanes
- Contra-rotating biplane Wells turbine
- Monoplane Wells turbine
- Biplane Wells turbine

A full-scale model of a Wells turbine with guide vanes has been installed in Japan (Mighty Whale project) and in The Azores (Pico pilot plant). The former featured eight NACA 0021 rotor blades and a power supply of 30 kW and the latter is a 500 kW rated plant.

Figure 1.1b) shows a variable pitch Wells turbine. The principle behind it is that depending on the approaching flow direction it varies the pitch angle between two predetermined angles. When the upstream flow hits the blade there’s a momentum transfer that results in a rotation of the blade about

![](image)

**Figure 1.1** a) Wells turbine with guide vanes; b) Wells turbine with variable pitch

![Image](image)

**Figure 1.2** a) Biplane Wells turbine with guide vanes; b) Contra-rotating biplane Wells turbine
an axis in the span wise direction located near the profile leading edge.

Contra-rotating biplane Wells turbine consists of two rotors placed next to each other rotating in opposite directions with no stator driving the flow. The downstream rotor is supposed to recover the swirl generated by the upstream rotor causing an axial flow when leaving the rotors. Such a turbine was first implemented at LIMPET plan (Islay, Ireland). The rotor had seven NACA 0012 profile blades and a power output of 500 kW. Another biplane type of turbine is the one where guide vanes drive the flow upstream and downstream from the rotors which both rotate in the same direction. This configuration was also applied at LIMPET plan.

A simple biplane Wells turbine features two rotor planes connected to the same shaft and therefore both with the same rotational velocity. Axial distance to blade chord ratio and upstream and downstream blades misalignment are parameters to take into account in such turbines.

A comparison between the first and the last three configurations have been made by means of an experimental test and the results are shown in [8]. It has been concluded that that contra-rotating and single-plane with guide vanes turbines have better performance of all and a similar performance between them with a slight advantage for the former in terms of efficiency and post-stall behavior.

![Figure 1.3](image)

**Figure 1.3** - a) Monoplane Wells turbine; b) Biplane Wells turbine

Regarding impulse turbines one can have:

- Self-rectifying impulse turbine with variable pitch guide vanes
- Self-rectifying impulse turbine with fixed guide vanes
- Contra-rotating McCormick turbine

Figure 1.4a) shows us a variable pitch angle self-rectifying impulse turbine. Similarly to the equivalent Wells turbine the guide vanes can rotate around a pivot placed near the edge located close to the rotor.
Pressure forces created by the flow as it passes by will drive the vanes in the right direction. This will do for a limited range rotation angles and will depend from which direction the fluid is coming from.

Such device was constructed by the National Institute of Ocean Technology in India. An impulse turbine will use fixed guide vanes to deflect the incoming flow both before and after the rotor. Because of the symmetric geometry of this configuration considerable losses are due when the fluid flow from the rotor to exit stator. However turbines of this type are intended to be constructed in China, Korea, India and Ireland.

A McCormick turbine is represented in Figure 1.5. It uses two contra-rotating rotors as well as two sets of fixed guide vanes before and after the first and second rotors respectively. A tested device has been reported to be very noisy.

Comparisons have been made [7] between some of the mentioned types of devices. It has been concluded that impulse turbines seem to have a better overall performance. Under irregular flow conditions, the ones real sea states provide, impulse turbines achieve better starting conditions as well as wider range of high efficiency values. Also lower rotation velocities are achieved which is a plus from the noise and mechanical point of view. However the Wells turbine isn’t put aside and still plays an important role among the wave energy power take-off systems.
1.5 Present Contribution

This work is a study of an alternative Wells turbine for wave energy harnessing. Such an exercise is done to judge if this device gathers acceptable operating characteristics so that its use can be justified. Several types of Wells turbines are known and analyzed. However this study is a preliminary one to a configuration that hasn’t been tested or which results haven’t been published. The latter is important because such a configuration has been patented already by Voith in 2006[9]. By the time this work was proposed to the author no information about any patent was had by the workgroup whatsoever. The intention was to evaluate what was believed to be an original idea. However after realizing the idea was no longer original, the interest on keeping with the works and its results was still maintained. A comparison is intended to be done with the “standard” turbine configuration with two stators and one rotor described in 1.4. The results will be compared directly to the ones described in [8]. The main idea behind it is that this type of turbine would be more robust from a structural point of view and less expensive to manufacture. The core of the stator would be stiff enough to store and bear a generator featuring a shaft ending for each one of the rotors. This advantage would also make it possible to decrease the lengthwise dimension of the turbine. Of course this would only be important if the efficiency of such a turbine was at least equivalent to the aforementioned term of comparison. And here lies the main intention of this paper: evaluating the aerodynamic performance of such a turbine. Three scaled-down prototypes were manufactured however only two were analyzed experimentally as well as from a 3D computational point of view. Also a monoplane Wells turbine was tested just to have an experimental term of comparison with old tests using the same rig. The experimental part is intended to evaluate efficiency, pressure drop and loss coefficients throughout the different stages of the two turbine configurations. Velocity profiles are also to be evaluated so that variations along radial direction can be accounted. Finally the CFD modeling reproduces the two tested turbines so that comparisons can be performed and analyzed.
2 Turbines Description

2.1 General Description

As mentioned before the turbine to be analyzed in this paper is new from the point of view of the location of its rotors and stator. The air to be pushed into the turbine in a real application scenario will face a first rotor. Downstream of the first rotor lays the stator. The stator will guide the air through its vanes so that the flow will enter the final set of rotor blades. The rotors are linked to each other through only one shaft. Consequently whatever the flow conditions are at each moment the model rotors will always have the same rotational speed. This geometry would allow a generator with two shaft endings to be assembled in the core of the turbine.

![Diagram](image)

Figure 2.1- "4_19_4" turbine model main dimensions and section overview. The stator blade shown does not reproduce the real geometry, being only representative. (All dimensions are in mm)

It is important to bear in mind that two turbine set ups are to be studied. The first one will have rotors with 4 blades each and twenty-nine vanes stator. The second one will feature rotors with four blades each and no stator in between. For the sake of simplicity from now on the former and the latter will be called "4_19_4" and "no-guide vanes" configurations respectively. Concerning the "no-guide vanes" model whereas in Figure 2.1 there are the stator blades there will be no guiding vanes instead. The hub radius \( r_{\text{hub}} \) and the casing radius \( r_{\text{casing}} \) are the same for all setups and 0.200 and 0.295m respectively.

It is also important at this time to define the coordinate system that will be valid throughout this paper. Moreover it will be introduced numbers that identify each intermediate stage of the turbine.
The presented coordinate system will be valid not only for the stator but also for the rotors. This can hence be considered the absolute inertial coordinate system.

In Figure 2.2, $\gamma$ stands for an arbitrary angle defined positive, $\theta$ represents the tangential direction and $x$ points in the axial direction.

2.2 Manufacturing process

The turbines were designed in such a way that the stators were built in the turbine structure being welded together with the casing and hub. This implied the existence of three different structures. All casing, hub and blades were made of Steel St37.2. Such material shows good stiffness and ductile behavior that makes it suitable for such manufacturing operations and application. Essentially casing and hub are made from sheets of steel, 6 mm each. Both casing and hub are connected through the stator blades. So that these blades can be placed in the correct position casing and hub sheets were cut with rips. These rips were cut before the steel sheets got roll-bended. This was done by a laser cutting process assisted by computer. This kind of cutting process exhibits very good precision characteristics. The rips will allocate the stator blade endings. Casing and hub steel sheets are bended in a roll forming process until they feature the desired radii. In the end the bended steel sheets are welded together so that the duct like structure can be obtained. Stator blades are manufactured in a similar way as casing and hub. A 2D plan drawing of the blade was made so that it

Figure 2.2- Turbine models in a x- $\theta$ plane

In Figure 2.2, $\gamma$ stands for an arbitrary angle defined positive, $\theta$ represents the tangential direction and $x$ points in the axial direction.
could be laser cut and roll bended afterwards (Appendix A2). The guiding vanes, 2 mm thick, in their final shape can therefore be inserted into the turbine through the radial direction into the casing until they reach the hub. Once one reaches this stage blades, casing and hub can be welded together piece by piece.

The no-guide vanes configuration has obvious differences in the way that no blades are present whatsoever. In order to link and uphold the casing pipe eight steel rods with a diameter of 10 mm were placed two by two in a 90° tangential distance.

Technical drawings for further details information are included at Appendix. This part of the manufacturing was done in an independent plant to which the job was ordered.

As mentioned before two different sets of rotors were used on the three different turbine setups. The entire rotor structure was made of Duralumin. This Aluminum alloy is a light weight alloy and shows good machinability characteristics. Some of the rotor related parts were manufactured at IST, Thermofluid Workshop. A complete rotor features three main components: the blades, a slotted ring, and two circular plates. The slotted ring is responsible for bearing the blades. For six bladed rotors this structure didn’t exist so it had to be manufactured as well as the circular plates. Both these parts were produced by means of turning and drilling operations. More specifically and in the case of the blade ring, the turning operation gave it the cylindrical shape whereas the drilling operation preformed the cuts that created the slots in which the blades will fit in.

2.3 The rotor

2.3.1 Rotor blades and Wells Turbine

Independent of the turbine configuration the blades mounted in the rotors had the same geometry. The profile used was NACA 0015. This code stands for the entity that created it, NACA (National Advisory Committee for Aeronautics), and it belongs to its four digit series of airfoils. The first digit indicates the maximum camber in percentage of the chord. The second one is a measure of the distance between the maximum camber coordinate and the leading edge (in tens of percent). The final two digits point out the maximum thickness as percentage of the chord. In this specific case NACA 0015 means that it’s a no camber profile and the total thickness is 15% of the cord.

Each “NACA XXXX” profile obeys a formula that will give the coordinates of the points that constitute the airfoil outline. The formula is a polynomial one and goes as follows[10]:

$$y = \pm \frac{t}{0.2} \left[ 0.2969 \sqrt{\frac{x}{c}} - 0.1260 \left( \frac{x}{c} \right) - 0.3516 \left( \frac{x}{c} \right)^2 + 0.2843 \left( \frac{x}{c} \right)^3 - 0.1015 \left( \frac{x}{c} \right)^4 \right] \quad (2.2)$$

where \( y \) is the vertical coordinate, \( x \) is the horizontal coordinate, \( t \) is the chord and \( c \) is the total profile chord

The above formula creates two lines that are symmetrical to each other about the chord line and will define both sides of the profile. Figure 2.3 shows a NACA 0015 looks like.
In the present configuration the blades have a 90° stagger angle. This feature identifies it with an already known type of turbine: the Wells turbine. Despite the high stagger angle a Wells turbine is an axial-flow turbine. It gained notoriety because its design allows the rotor to turn always in the same direction independent of the direction of the incoming air flow. Such behavior becomes particularly important on an OWC plant. Due to its oscillating flow pattern, the air that is pushed and sucked into the turbine will approach the blades from opposite sides. However rotor, shaft and generator will rotate in only one direction and it can produce a time-averaged positive power. Due to the fact of permitting incoming axial-flow from either side it is classified as a self-rectifying turbine.

Let us now look at the velocity triangle and force diagram of a single blade shown in Figure 2.3:

![Figure 2.3- Wells turbine velocity triangle and forces diagram](image)

It is important to refer that in Figure 2.3 the subscripts 1 and 2 refer to upstream and downstream of a theoretical single blade in this particular example and no relation exists with the notation introduced in 2.1 (Figure 2.1).

Assuming an upstream axial absolute velocity $C_1$ a fluid particle will face a moving blade with a perpendicular velocity $U$. In a blade cascade the relative resultant velocity vector $W_n$ will form an angle $\beta$ with respect to the axial direction. $W_n$ will generate two forces on the airfoil: one perpendicular and another tangent to its direction. Those are formally called Lift and Drag and represented by $L$ and $D$. If one now decomposes both $L$ and $D$ in its tangential and normal components relative to the chord one gets two forces: $F_u$ and $F_n$ respectively. $F_u$ will be the force driving the rotor in its rotational motion. The right balance between $L$ and $D$ will be crucial for the resultant $F_u$. A suitable angle of attack $\beta$ and relative velocity magnitude for the cascade $W_n$ will generate a positive $F_u$. However this may not happen. When large drag forces are generated they will induce a negative $F_u$ which of course is the non-desired situation. Drag becomes dominant when $\beta$ is close to zero or when it overcomes a limit value above which the Lift forces suddenly decrease. This last situation is called stalling and is caused when large angles of attack are imposed to an airfoil.

If one now imposes the exact same conditions but with an upstream axial velocity coming from the opposite blade direction the result will be the same meaning that the rotational speed will be in the same direction.
2.3.2 Rotors for “4_19_4” turbine model and “no guide vanes” configurations

As mentioned before two different sets of rotors were manufactured. The main difference between them is the assembled number of blades. However only the four bladed rotors were assembled into the model and tested afterwards. The rotor is the entire part that includes the blades and the structure that bears them. For such reason it can also be called a bladed disk. A “bladed disk” is one in which the blades are separated parts and are assembled one by one. In this particular case the “disk” is a ring instead. The slotted ring is responsible for properly fixing the blades. The coupling between the blade roots and the ring is achieved through fir-tree look like slots. The latter constrains the detachment of blades and ring in the radial direction. Some damping is also achieved by such a link between parts. To axially fix the blades roots two circular plates act as stoppers in each side of the ring, being bolted to it. One of these plates is then fixed to the shaft through a structure that ensures the concentricity of shaft and rotor.

All blades have constant chord and untwisted profiles in the spanwise direction. The rotors had all an outer diameter of 0.59m.

2.4 The guide vanes

The design of the stator guide vanes and its methods lie outside the essence of this work. However a brief explanation about such matter is important to understand how the design process was driven. Since the beginning the intention was to design guide vanes that could be, at the same time, easy to make from a manufacturing point of view as well as cost-effective. Therefore the solution achieved was a stator blade with constant thickness. The latter would allow the guiding vanes to be made of thin sheets of steel that are cheap and easy to work on at the same time. The result predicted a blade geometry in which the profile curvature would vary in the span wise direction. However and once again for the sake of manufacturing simplicity it was decided to maintain a constant curvature through the radial direction. The guide vanes were designed based on Zweifel criterion and Weinig theory. The Zweifel criterion defines the optimum pitch to chord ratio so that friction losses can be minimized and flow guidance can be optimized. Weinig theory allows a potential flow analysis for a cascade profile geometry. This is a potential flow analysis and a conformal transformation can be done to get the real guiding vanes geometry. The main design parameters that resulted from the aforementioned design constraints can be seen in Appendix Table 1. As one can observe the stator is symmetrical. This means that both inlet and outlet blade angles are have the same absolute value and opposite directions in respect to the axial direction. According to [11], by making a 2D potential flow analysis the absolute flow angle downstream of a Wells blade row will only be dependent of the upstream absolute flow angle upstream of the blade row and its solidity. This applies for fixed rotational velocity and flow rate. This means that in theory the flow deflection on the first and second rotors would be the same. This justifies the geometrical symmetry of the stator.

To properly understand parameters from Table 2.1 let us first define a non-dimensional parameter that is a function of a radial coordinate (r):
From the above one should realize that a coordinate with $R^* = 0$ means that it is located at the hub surface whereas a point with $R^* = 1$ is placed at the casing surface. Logically $R^* = 0.5$ will indicate a position halfway between the hub and casing.

### 2.5 Working and thermodynamic basic principles

As a self-rectifying turbine the rotation direction of its rotors will be always the same. Its rotational velocity that can directly drive electrical generators that feed the electrical grid. Such velocities can also induce the rotor to create the so-called fly-wheel effect. This is the capability of a rotor maintaining the rotation even if no flow is going through the turbine. Combining mass moment of inertia and velocity, the stored kinetic energy can keep on spinning the rotor. However such rotational velocities and the resultant flow field won't lead one to take into consideration the compressibility effects of the air. Compressibility effects are important whenever the Mach number becomes larger than 0.3. Since an incompressible analysis is to be performed no variations of air density will be considered. Therefore in such an analysis it is only necessary to define air density, $\rho$, and kinematic viscosity, $\mu$, to fully define the working fluid.

Even though blades stagger angle is very large this turbine is still an axial turbine. Axial turbines are the ones where the flow pattern is considered to have close to zero radial velocity over the stream wise direction. Hence the velocity field will have two major components: axial and tangential. The incoming flow is considered to be purely axial. This enables direct calculation of the mass flow rate over which the turbine is running. Mass flow rate is defined as:

$$m = \rho Q = \rho C_x A$$

where, $\rho$ is the air density (kg/m$^3$), $Q$ is the flow rate (m$^3$/s), $C_x$ is an arbitrary axial velocity (m/s) and $A$ is the turbine annular cross section (m$^2$)

This quantity is conserved through the turbine according to the mass conservation principle.

Figure 2.4 shows the velocity triangles over the turbines different stages from section 0 to 3. These 2D diagrams are representations of what would be the behavior of the fluid particles when going through a spatially averaged pattern. Three different velocities are considered in these diagrams: $C$- absolute velocity; $W$- relative velocity; $U$- rotor tangential velocity. Indexes $x$ and $\theta$ represent the axial and tangential components respectively.
Absolute velocity is the one represented in a fixed reference frame. Rotor tangential velocity is calculated according to the following formula:

\[ U = \Omega r \]  \hspace{1cm} (2.4)

Where \( \Omega \) is the rotor angular velocity (rad/s) and \( r \) is the distance to the axis of rotation (m).

Finally \( W \) is the relative velocity between the aforementioned velocities and is the one seen from the rotational reference frame located in the rotor while rotating. The angles \( \alpha \) and \( \beta \) stand for absolute and relative flow angles respectively and are both defined relatively to the axial direction.

In section 0 the absolute velocity vector only has an axial component. The combination of the latter with the rotor rotational velocity will create \( W_0 \). The direction of \( W_0 \) is of paramount importance to this first stage of the turbine. A proper resultant \( W_0 \) (from \( C_0 \) and \( U \)) will largely affect the efficiency of this first stage in the way that they will dictate the short range of attack angles (\( \beta_0 \)) on which stall does not occur for the rotor blade profile.

At this point it becomes relevant to define non-dimensional parameter that will be widely mentioned from now on. It is called flow coefficient and is defined as:

\[ \varphi = \frac{C_x}{U} \]  \hspace{1cm} (2.5)

This parameter can be a direct measure of flow-rate, velocity triangles height and first rotor angle of attack.
When leaving the first rotor a deflection has been induced to the flow field resulting in tangential absolute velocity component $C_{\theta 1}$.

In case of "4_19_4" configuration the absolute velocity direction at rotor exit should in principle be the same as the blade metal angle so that losses can be minimized. As mentioned before the stator is symmetric relative to an $\mathbf{r}$-$\theta$ plane intersecting the stator at the maximum camber point. Therefore the deflection caused by the rotor results in an angle $\alpha_2$ that is symmetric to $\alpha_1$ with respect to the axis of rotation. The next stage is the entry to the second rotor. Once again combining the absolute flow velocity resultant from stator exit and the rotational velocity of the rotor, a relative velocity ($W_2$) will face the second and final rotor and will push the second rotor to rotate. The final section 3 is one where the flow freely leaves the turbine. According to [12] if one reduces the outflow swirl one is promoting less kinetic energy losses and therefore it will lead to improved efficiency. The absolute velocity is intended to have none tangential absolute velocity component and hence $\alpha_3=0^\circ$. As a result of a high-efficiency flow coefficient the outflow should, in theory, be axial. In order to achieve this last condition a correct design of stator vanes is of considerable importance as it drives the flow to the final rotor.

In the no-guide vanes configuration the flow leaving the first rotor won't be deviated by any guide vanes and the absolute direction that a flow particle features will be the one facing the second rotor. This means that velocity triangles in sections 1 and 2 in the no-guide vanes turbine have the same velocity vectors and therefore $C_{\theta 1}=C_{\theta 2}$, $W_{\theta 1}=W_{\theta 2}$. The reason for not attributing one different number or letter to this section is to maintain notation simplicity and also to corroborate the next definitions and formulas that are valid for both configurations. In such a turbine the flow will never experience a close to zero swirl at turbine outlet.

![Velocity Triangles](image.png)

Figure 2.5- "No guide vanes" turbine velocity triangles
If an energy balance is applied to an arbitrary rotor and considering that no heat transfer occurs between the turbine and the surroundings (adiabatic conditions) it can be shown that:

\[ \dot{W} = \dot{m} \Delta h_0 \]  

(2.6)

In the above formula \( \dot{m} \) is the mass flow rate (kg/s) and \( \Delta h_0 \) is the total enthalpy difference in (J/kg). Applying (2.6) to our case it comes:

\[ \dot{W} = \dot{m}[(h_{00} - h_{01}) + (h_{02} - h_{03})] \]  

(2.7)

But if one makes use of Euler's turbine equation, total enthalpies can be replaced by velocities as follows:

\[ h_{00} - h_{01} = U (C_{00} - C_{01}) \]  

(2.8)

\[ h_{02} - h_{03} = U (C_{02} - C_{03}) \]  

(2.9)

\( U \) remains the same for both rotors since they're linked through the same shaft and \( C_{00} \) is assumed to be zero yielding to:

\[ \dot{W} = \dot{m} U [(-C_{01}) + (C_{02} - C_{03})] \]  

(2.10)

Since the stator is a non-rotating component Euler's equation will result that total enthalpy is conserved and therefore one has:

\[ h_{01} - h_{02} = 0 \]  

(2.11)

It is now important that one defines a very important parameter for any type of turbomachine- its efficiency. According to the experimental nature of this work the efficiency to be considered is the hydraulic efficiency, also called total-to-total efficiency. From a thermodynamic point of view it is the ratio between the energy actually transferred between the air and the turbine and the energy transfer that would occur if the process was isentropic (no losses). Aerodynamic losses will be accounted for in every stage of the turbines and a thorough analysis is due on Chapter 4.
3 Experimental Analysis

The experimental facility is located at the Engineering Mechanics Department at IST. The main goal of the experiments was to evaluate efficiencies as well as velocity triangles and pressure drop. To make this happen, a proper experimental rig had to be set up. The experimental rig includes all main components of the turbine as well as all the necessary instruments that allow the evaluation and measurement of the important variables in question.

3.1 Experimental rig

The experimental rig includes three different major parts: the turbine itself, the plenum chamber and finally the fan. Figure 3.1 shows the relative position of these major components as well as pressure tapping’s locations.

Hub and casing diameters is 0.4 and 0.59 m respectively and kept constant through rotors and stator sections. Assembled inside the turbine hub are the torque transducer and a Siemens motor/generator. This generator can act as motor or generator depending on the turbine flow conditions and its rotational speed is self-tuned to be kept constant. Both generator and torque transducer are linked in line with the turbine shaft. The turbine includes a convergent nozzle that helps driving the air from the surroundings into the turbine.

In between the last rotor and the stagnation chamber lays a wood diffuser. The stagnation chamber is a box-like structure of about 2 m long, 1.72 m height and 1.78 m width. Inside exists a honey lattice made of 30 mm diameter pipes separating the two sides of the chamber. The downstream side of the chamber is connected with a fiber glass calibrated convergent nozzle. Its wider section features a
diameter of 0.8 m that gradually decreases to a 0.4 m diameter section over a length of about 0.67 m. Last there's a centrifugal fan responsible for sucking the air into the turbine. The air flow rate across the turbine can be controlled by adjusting the rotational velocity of the fan. Throughout the turbine major parts there are pressure tappings located in strategic places (from section 1 to 3). Being tangentially equi-spaced in each annular measuring section they are also equally spaced over the sides of the plenum chamber (section 4). The pressure tappings would be afterwards connected in parallel so that an average pressure over each section can be read by the pressure manometers.

![Figure 3.2- Turbine model](image)

### 3.2 Instrumentation

Essentially three different properties need to be measured: rotor angular velocity, shaft torque and pressures (static and total).

Pressure measurements throughout the turbine are performed by means of five pressure manometers. Those were manufactured by Furness Controls Limited being the FC012-Micromanometer model. Such devices have as input pneumatic pressures that can be read through two pressure tappings at each manometer. The pressures are read in mmH₂O are then transformed into voltage consisting of an output analological DC signal of ±5 V. Three different pressure range manometers were used. However each manometer can read 100% or 10% of its maximum value. Depending on the section pressures are being measured different scales are due. Table 3.1 resumes the maximum value that can be read in each scale according to the different manometers used.

<table>
<thead>
<tr>
<th>Manometer</th>
<th>Main scale (mmH₂O)</th>
<th>Secondary scale (mmH₂O)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1999.9</td>
<td>199.9</td>
</tr>
<tr>
<td>2</td>
<td>199.9</td>
<td>19.99</td>
</tr>
<tr>
<td>3</td>
<td>199.9</td>
<td>19.99</td>
</tr>
<tr>
<td>4</td>
<td>199.9</td>
<td>19.99</td>
</tr>
<tr>
<td>5</td>
<td>19.99</td>
<td>1.999</td>
</tr>
</tbody>
</table>

Table 3.1- Pressure manometers scales
As mentioned before there's a torque transducer assembled in line with the motor/generator responsible for driving the shaft. The torque transducer is a device that has to shaft endings one connected to the generator and the other one connected to the rotor shaft. The relative twist between these two endings will dictate the torque applied to the rotor. It's the VIBRO-METER TG-5/BP model and can read a maximum torque value of 50 Nm. For this application this value is suitable even for torque peaks that might occur during some velocity variation. The output is an analogical voltage of ±10 V DC signal.

Attached to the torque transducer is a photoelectric cell, VIBRO-METER ML103, responsible for measuring the angular velocity of the shaft. It is directed into a gear-wheel in which the cogs will activate the cell making it possible to calculate the rotational velocity by pulsing electrical signals in every revolution.

Both torque transducer and photoelectric cell are connected to power supply unit, model VIBRO-METER APD155, through two cables. These cables feed the components and the same time they carry their signals. This unit also transforms the photoelectric cell pulsing electric signals into DC current and amplifies the torque signal.

To determine the flow angles a transverse moving probe was used. This probe has two degrees of freedom. It moves up and down in the radial direction and it can also rotate around its own axis. This displacement is controlled by its own unit in which both radial position and yaw angle are indicated. The probe has three total pressure tappings (PT 7, 8 and 9) all lined up as shown in Figure 3.2.

![Transverse probe assembled on turbine model](image)
3.3 Data acquisition

The analogic signals coming from the all the devices described in 3.2 converged to an acquisition data card DAQCard-6024E from National Instruments. The commercial software LabView was the platform to gather and treat the experimental data. This platform allows, among others, to have an online control of the parameters one wishes to measure. This enables the user to better judge when and if the flow parameters signals are ready to be acquired. Three main routines were used in order to calibrate the transverse probe, get the efficiency curves and the flow angles.

3.3.1 Efficiency, pressure drop and torque coefficients

In order to evaluate efficiencies several curves were obtained for a constant rotational velocity. The procedure is such that one sets a specific rotor angular velocity and then by varying the power of the fan gradually decreases the stagnation pressure inside the plenum chamber. This pressure drop will induce a certain flow rate across the turbine. For each rotational velocity and flow rate pair of values several flow parameters are recorded: pressures at PT1, PT2, PT3 and PT4; torque; rotational velocity and flow rate. Turbine pressure tappings from section 1 to 4 are connected each one to its own pressure transducer. In these cases read values of pressure are always relative to the atmospheric pressure. The flow rate is calculated by using the pressure difference between PTi and PTii. By the means of a Bernoulli equation the volumetric flow rate (m³/s) was calculated as follows:

\[
Q = A_n \frac{2}{\sqrt{\rho k}} \Delta p_{lii}
\]  

(3.1)
Where $A_n$ is the cross sectional area of the convergent nozzle ($m^2$), $\rho$ is the air density, $k$ is a specific nozzle flow loss coefficient which is equal to 0.953 and $\Delta p_{i,ii}$ is the static pressure difference between section PTi and PTii represented in Figure 3.1;

Five pressure transducers were used for the efficiency tests for the 19 blades stator configuration and four pressure transducers were used for the “no guide vanes” configuration.

Different angular velocities from 1200 to 2750 rpm were set to get the efficiency curves. According to the similarity laws and being a non-dimensional parameter it implies that all the curves for the different constant rotational velocities will be mapped on top of each other.

The hydraulic efficiency is calculated by means of the following formula:

$$\eta = \frac{(T_{\text{aer}})\omega}{Q\Delta p_0}$$  \hspace{1cm} (3.2)

where,

$$T_{\text{aer}} = T + T_{\text{losses}}$$ \hspace{1cm} (3.3)

Being $T_{\text{aer}}$, the torque generated by the aerodynamic forces (N.m), $T$ is the total torque read by the torque transducer (N.m), $T_{\text{losses}}$ the torque resultant from friction forces on the bearings (N.m), $\omega$ is the rotors angular velocity (rad/s), $Q$ is flow rate ($m^3$/s), $\Delta p_0$ is the total pressure drop meaning that is the total pressure difference between section 0 (assumed to be 0 Pa) and section 4 (stagnation chamber, Pa).

The signal acquisition frequency was of 100 Hz and the number of samples was 1000, meaning that the final value for each measured property is a time averaged value over a period of 10 seconds.

One should also define other dimensionless parameters such as torque coefficient and total pressure drop coefficient. These become important to analyze the overall performance of the turbines and are defined respectively as follows:

$$T^* = \frac{T_{\text{aer}}}{\rho \omega^2 R^2}$$  \hspace{1cm} (3.4)

$$\Delta p_0^* = \frac{\Delta p_0}{\rho \omega^2 R^2}$$  \hspace{1cm} (3.5)

Where for both cases $R$ is outer radius ($R = 0.295m$).

In order to evaluate the friction forces generated inside the bearings some modifications had to be performed to the turbine. The entire turbine structure was made from scratch and thus a friction torque ($T_{\text{losses}}$) analysis was due. So that no aerodynamic forces can be generated, the blades had to be removed from the rotor. The next step was to impose several rotational velocities to the rotor and measure the resultant torque. The value read by the torque transducer would be the friction torque that varies according to the rotor angular velocity.
By doing this kind of evaluation it is possible to calculate the aerodynamic torque generated by the rotors and correct the torque measured by the torque transducer for every specific angular velocity the turbine is running.

### 3.3.2 Probe Calibration

The three holes transverse moving probe is the device that measures the flow angles as well as total pressures in the different sections where it will be used. In order to help fixing the probe to the structure that will set the radial position and yaw angle a spirit level is assembled on the probe. The goal is to align the spirit level so that the holes can all be directed towards the axial direction. However a deviation angle \( \alpha_d \) between the spirit level and holes is present and should be determined. Also a relation between the pressures measured at each hole and the total and dynamic pressures was necessary. To find all these relations a calibration test was performed.

To calibrate the probe it was necessary to impose a flow direction that could be maintained constant and known. The right way to do it is to place the probe on a wind tunnel. In fact the probe was located at the wind tunnel opening, meaning that it was just “capturing” the free-stream right outside the wind tunnel channel. It is assumed that velocity profile is kept constant and the static pressure is atmospheric pressure. Along with the probe a Pitot tube was also placed at the same section so that total pressure could be measured. Five pressure manometers were used for the calibration. Three for each hole of the transverse probe (PT 7, 8 and 9), one for the Pitot tube and one other to measure the pressure difference between holes 7 and 9. All pressure transducers but the latter were measuring the pressure with respect to atmospheric pressure. Therefore the Pitot tube total pressure matches, in this case, the dynamic pressure. The presence of the Pitot tube is justified by being the reference in terms of total pressure so that a relation between the pressures in the three holes can be related with the local total pressure.

One should now define the calibration coefficients for the flow angle, total and dynamic pressure:

\[
C_\alpha = \frac{p_7 - p_9}{p_8 - \bar{p}_{7-9}} \\
C_{tot} = \frac{p_8 - p_{tot}}{p_8 - \bar{p}_{7-9}} \\
C_{dyn} = \frac{p_8 - \bar{p}_{7-9}}{p_{dyn}}
\]

Being,

\[
\bar{p}_{7-9} = \frac{p_7 + p_9}{2}
\]
After setting the ROTADATA control unit close to the wind tunnel the procedure was to acquire the pressures on the Pitot tube and probe holes for each yaw angle. The range of yaw angles was from -10 to 10º with a step of 1º meaning that 21 points were measured. The air velocity at the test section was kept constant about 12.7 m/s.

Ideally when the probe is aligned with the flow the pressure on pt 8 should be the total pressure and the difference between p7 and p9 should be zero. By doing such a calibration one can get the relations between the aforementioned coefficients and the deviation angle $\alpha_d$.

### 3.3.3 Flow angles

After the calibration of the transverse probe was done the probe could finally be assembled on the turbine. The control unit that bears the probe lies on a platform that assures its radial position. In the beginning of each test the probe is placed with its pressure taps close to the hub. The experiment runs with increasing radial positions so that flow angles and total pressure can be read for different radial coordinates. The program that controls the probe is such that only the initial position, range of angles over which that flow might be coming from and maximum pressure difference between probe lateral wholes are asked. The routine in charge of the measurements runs on its own and is based on the following iterative process:

1. The probe is set on the initial position $\alpha_A$ and the pressure difference $\Delta p_A = p_7 - p_9$ is registered.
2. The probe will rotate to a new position in a direction that evens the pressure difference between $p_7$ and $p_9$.
3. Being on the new position, $\alpha_B$, the probe will evaluate again the pressure difference on the lateral pressure tappings, $\Delta p_B$.
4. Based on the pair of points $((\Delta p_A, \alpha_A), (\Delta p_B, \alpha_B))$ on the last two iterations a subroutine will apply a linear interpolation in order to find a new position $\alpha_C$, based on the following formula:

   $$\alpha_C = \alpha_A + \frac{(\alpha_B - \alpha_A)}{\Delta p_A - \Delta p_B} \Delta p_A$$

   (3.10)

5. If the new angle is larger or smaller than the maximum and minimum angle imposed previously the final angle will be one of the last two, more specifically the one that is closer to the actual position. Otherwise the new angle is given by $\alpha_C$.
6. The program reads the pressure values on the three pressure tappings.
7. If the new pressure difference $p_7 - p_9$ is smaller than the stipulated tolerance than the angle value is registered along with the pressures on the each hole and the respective radial position and the prove increments its radial position. In contrary the program will use the formula written in point 4 to calculate the new yaw position based on the last pair of points B and C.
8. In case the maximum number of allowed iterations is exceeded the actual values are recorded and the probe moves to a new radial position.

For these experiments five pressure transducers were used. Three for each probe hole, one to measure the pressure difference between PT7 and PT9, and one other to calculate the flow rate as described before.

Regarding the "4_19_4" configuration the flow angles were evaluated in three different sections. The first one was between the first rotor and the stator. The second one was between the stator and the second rotor. On this particular one the measurements were done in more than one tangential position. Five different tangential locations were evaluated so that a complete velocity profile over one stator pitch could be acquired. The first and final tangential positions are located one pitch away from each other. This means that the flow pattern along the radial coordinate should be very similar according to the periodicity of the total flow. Finally the last transverse experiments were performed downstream the last rotor. Since downstream the rotors the flow is mixed up there was no need to perform tests in more than one point.

On the “no-guide vanes” configuration transverse probe measurements were made only on two sections both downstream the two rotors. The first one half-way between the rotors and the other about one rotor blade cord length downstream the second rotor.
4 Experimental Results

4.1 Transverse probe calibration

The results that arise from the probe calibration can be seen in the following plots. Figure 4.1 shows the pressures read by the probe relative to the total pressure measured by the Pitot tube. One can observe that $p_9/\text{ptot}$ and $p_7/\text{ptot}$ have inverse behaviors as one passes from negative to positive angles. This is due to the fact that since they’re on opposite sides of the probe one of them will be always more closely directed towards the incoming flow and therefore capturing a higher pressure. The parameter $p_8/\text{ptot}$ remains a fairly constant line close to 1. The pressure difference $p_7-p_9$ relative to the total pressure doesn’t intersect the origin of the coordinate axis revealing the existence of a deviation angle between the spirit level and the probe. The deviation angle can be determined by adjusting trend lines to the plots on Figure 4.2.

If ones makes use of the trend line equation for $L_x$ and calculate where it intersects the $\alpha$ axis one can get the deviation angle $\alpha_d$. In this case $\alpha_d = 6.26^\circ$. Once this angle is known it will be added to the one the probe controller digitally indicates since when the spirit level is centered it corresponds to an angle of 0$^\circ$ in the controller.

After knowing the flow angle with the respective correction $p_{\text{tot}}$ and $p_{\text{dyn}}$, can be determined by using equations 3.7 and 3.8. At this point one has the necessary elements to calculate both direction and intensity of the velocity vectors.
The friction torque evaluation test is important to correctly measure the aerodynamic torque according to 3.3. The friction torque is a parameter that varies with the shaft rotational velocity.

Two experiments regarding friction torque were run: one where friction torque was measured at turbine cold start and one other when turbine has been running for half an hour. The reason for such is that bearings tend to offer less rolling resistance when warmed up. The results proved the latter true as can be seen in Figure 4.3.

It can be seen that friction torque increases with increasing rotational velocity. A trend line was applied and friction torque will be assumed to vary according to table 4.2:

<table>
<thead>
<tr>
<th>Trend line equation</th>
<th>R²</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{\alpha} = -0.081\alpha + 0.507$</td>
<td>0.999</td>
</tr>
<tr>
<td>$C_{\text{dyn}} = 0.0001\alpha^2 + 0.534$</td>
<td>0.927</td>
</tr>
<tr>
<td>$C_{\text{tot}} = 0.00001\alpha^2 + 0.006\alpha - 0.057$</td>
<td>0.993</td>
</tr>
</tbody>
</table>

Table 4.1- Calibration coefficients trend line equations

### 4.2 Friction torque

The friction torque evaluation test is important to correctly measure the aerodynamic torque according to 3.3. The friction torque is a parameter that varies with the shaft rotational velocity. Two experiments regarding friction torque were run: one where friction torque was measured at turbine cold start and one other when turbine has been running for half an hour. The reason for such is that bearings tend to offer less rolling resistance when warmed up. The results proved the latter true as can be seen in Figure 4.3.

It can be seen that friction torque increases with increasing rotational velocity. A trend line was applied and friction torque will be assumed to vary according to table 4.2:
In order to account for friction torque losses the trend line for “after 30 min” was the chosen one, given that it simulates steady conditions in a better way. It is also important to mention that all the experiments were made after a turbine warming up period of time.

<table>
<thead>
<tr>
<th>Trend line equation</th>
<th>$R^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cold start ( T_{\text{losses}} = 2,5 \times 10^{-7} \Omega^2 - 5,5 \times 10^{-4} \Omega + 6,1 \times 10^{-1} )</td>
<td>$6,1 \times 10^{-1}$</td>
</tr>
<tr>
<td>After 30 min ( T_{\text{Losses}} = 1,8 \times 10^{-7} \Omega^2 - 2,1 \times 10^{-4} \Omega + 1,4 \times 10^{-1} )</td>
<td>$8,8 \times 10^{-1}$</td>
</tr>
</tbody>
</table>

Table 4.2 - Friction losses trend line equations

4.3 Efficiency, pressure drop and torque coefficients

The efficiency tests show the performance of the two turbine configurations under different flow coefficients. Plots of the efficiency curves can be analyzed on Figure 4.4 and Figure 4.7. The curves that resulted from each angular velocity fairly lie on top of each other.

4.3.1 “4_19_4” turbine

The efficiency curves that resulted from the different rotational velocities are mapped on top of each other. This is a good indicator that the experiments were well run. Flow coefficient values vary from 0.09 to 0.32.

One can observe a peak efficiency of about 64% for a flow coefficient of about 0.2. Figure 4.4 also shows that there’s a considerable range of flow coefficients, from 0.15 to 0.25, over which the efficiency remain close to its peak value.
The total pressure drop coefficient seems to vary linearly with the flow coefficient from 0.14 to 0.42.

Figure 4.6 shows the torque coefficient curves for different rotational velocities. It can be observed that torque coefficient seem to increase with the flow coefficient up until stall occurs. After that there’s a sudden plunge that result in a low turbine efficiency.

### 4.3.2 “No guide vanes” turbine

For the “no guide vanes” turbine the curves again lie very close to each other. Some irregularities are noticeable for low flow coefficients (also low power outputs) that can be justified by the fact that for such a range of torque values the efficiency (and torque coefficient) curve become very sensitive to the friction torque correction made afterwards the torque data is read by the torque transducer. Disregarding the peak efficiency for low flow coefficient there’s a maximum efficiency values around $\phi=0.2$ ($\eta=0.67$). Efficiency remains around its maximum value from $\phi=0.17$ to $\phi=0.28$. After this efficiency curve drops to very low values.
Total pressure drop vary linearly up until $\phi = 0.27$, after that it keeps increasing less rapidly. Its values vary from approximately 0.12 to 0.52.

When it comes to torque coefficient its variation is fairly linear until stall condition. For this flow coefficient ($\phi = 0.28$) torque coefficient reaches its maximum ($T^* = 0.11$).
4.3.3 Error analysis

When performing an experimental analysis one should be bear in mind that there is always uncertainty intrinsic to the acquisition data process and its numerical manipulation. Consequently error analysis becomes important and acts as a measure of reliability one can have on the results. According to [13] errors may be divided into two main groups: bias limit and precision limit errors. The former are also known as systematic errors and may arise from bad calibration of the instruments, inadequate conduction of the experience or even low instrument accuracy. This can lead to a low accuracy in the results, i.e. considerable distance between real and measured quantities. Precision limit errors, also known as random errors, are intrinsic to the measuring process. These can come from electronic fluctuation or reading error, both are out of observer influence.

The uncertainty of a measurement is affected by both bias limit and precision errors. It can be determined by:

$$u = \sqrt{b^2 + p^2}$$  \hspace{1cm} (4.1)

Where $b$ and $p$ are bias limit and precision error respectively.

By calculating the uncertainty one can assume that the true value of a property $x$ will lie within the interval $[x - u, x + u]$ with 95% of confidence. Bias limit means that the one running the experiment is 95% certain that $b$ is inferior to the real systematic error. If this error is introduced by $K$ different components it will come as:

$$b = \sqrt{\sum_{k=1}^{K} (b_k^2)}$$  \hspace{1cm} (4.2)

However it can be more simply estimated by reading instrumentation manufacturer’s specifications. Table 4.3 shows the several bias errors provided by the manufacturers of the acquisition data hardware used for the experiments.

Figure 4.9- Torque coefficient for several angular velocities

<table>
<thead>
<tr>
<th>$T^*$</th>
<th>1300 RPM</th>
<th>1500 RPM</th>
<th>1750 RPM</th>
<th>2000 RPM</th>
<th>2200 RPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.108</td>
</tr>
<tr>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
<td>0.216</td>
</tr>
<tr>
<td>0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.324</td>
</tr>
<tr>
<td>0.06</td>
<td>0.06</td>
<td>0.06</td>
<td>0.06</td>
<td>0.06</td>
<td>0.432</td>
</tr>
<tr>
<td>0.08</td>
<td>0.08</td>
<td>0.08</td>
<td>0.08</td>
<td>0.08</td>
<td>0.540</td>
</tr>
<tr>
<td>0.10</td>
<td>0.10</td>
<td>0.10</td>
<td>0.10</td>
<td>0.10</td>
<td>0.648</td>
</tr>
<tr>
<td>0.12</td>
<td>0.12</td>
<td>0.12</td>
<td>0.12</td>
<td>0.12</td>
<td>0.756</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$T^*$</th>
<th>1300 RPM</th>
<th>1500 RPM</th>
<th>1750 RPM</th>
<th>2000 RPM</th>
<th>2200 RPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.108</td>
</tr>
<tr>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
<td>0.216</td>
</tr>
<tr>
<td>0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.324</td>
</tr>
<tr>
<td>0.06</td>
<td>0.06</td>
<td>0.06</td>
<td>0.06</td>
<td>0.06</td>
<td>0.432</td>
</tr>
<tr>
<td>0.08</td>
<td>0.08</td>
<td>0.08</td>
<td>0.08</td>
<td>0.08</td>
<td>0.540</td>
</tr>
<tr>
<td>0.10</td>
<td>0.10</td>
<td>0.10</td>
<td>0.10</td>
<td>0.10</td>
<td>0.648</td>
</tr>
<tr>
<td>0.12</td>
<td>0.12</td>
<td>0.12</td>
<td>0.12</td>
<td>0.12</td>
<td>0.756</td>
</tr>
</tbody>
</table>
When a property $y$ is derived from a mathematical expression where all the parameters are independent, the bias error will be calculated as follows:

$$ b_y = \sqrt{\sum_{m=1}^{M} \left( \frac{\partial y}{\partial x_m} b_{x_m} \right)^2 } $$  \hfill (4.3)

Similarly to the bias error, the precision limit error signifies that there is 95% probability of this error being inferior to the real precision limit error. When there is a considerable amount of repeated measures of the same property over the same circumstances it can be assumed that measurements follow a Gauss distribution. The precision limit error is calculated by:

$$ p = 2\sigma_x $$  \hfill (4.4)

Where $\sigma_x$ is the standard deviation of the set of measurements. If a specific property $y$ is calculated using $x_m$ independent variables the precision error will be:

$$ p_y = \sqrt{\sum_{m=1}^{M} \left( \frac{\partial y}{\partial x_m} p_{x_m} \right)^2 } $$  \hfill (4.5)

Applying the described theory to determine the flow rate uncertainty one will use:

$$ b_Q = \sqrt{\left( \frac{\partial Q}{\partial k} b_k \right)^2 + \left( \frac{\partial Q}{\partial \Delta p} b_{\Delta p} \right)^2 } $$  \hfill (4.6)

$$ p_Q = \sqrt{\left( \frac{\partial Q}{\partial \Delta p} p_{\Delta p} \right)^2 } $$  \hfill (4.7)

$$ u_Q = \sqrt{b_Q^2 + p_Q^2 } $$  \hfill (4.8)

To determine efficiency uncertainty one makes use of the following formulas:

$$ b_\eta = \sqrt{\left( \frac{\partial \eta}{\partial T} b_T \right)^2 + \left( \frac{\partial \eta}{\partial w} b_w \right)^2 + \left( \frac{\partial \eta}{\partial Q} b_Q \right)^2 + \left( \frac{\partial \eta}{\partial \Delta p} b_{\Delta p} \right)^2 } $$  \hfill (4.9)

$$ p_\eta = \sqrt{\left( \frac{\partial \eta}{\partial T} p_T \right)^2 + \left( \frac{\partial \eta}{\partial w} p_w \right)^2 + \left( \frac{\partial \eta}{\partial Q} p_Q \right)^2 + \left( \frac{\partial \eta}{\partial \Delta p} p_{\Delta p} \right)^2 } $$  \hfill (4.10)
Similarly the torque coefficient uncertainty is given by:

\[
\begin{align*}
    b_\tau^* &= \sqrt{\left(\frac{\partial T^*}{\partial T} b_T\right)^2 + \left(\frac{\partial T^*}{\partial b} b_w\right)^2} \\
    p_\tau^* &= \sqrt{\left(\frac{\partial T^*}{\partial T} p_T\right)^2 + \left(\frac{\partial T^*}{\partial p} p_w\right)^2} \\
    u_\tau^* &= \sqrt{b_\tau^* + p_\tau^*}
\end{align*}
\]  

Finally the pressure drop coefficient uncertainty is derived using the following expressions:

\[
\begin{align*}
    b_{\Delta p_0^*} &= \sqrt{\left(\frac{\partial \Delta p_0^*}{\partial \Delta p_0} b_{\Delta p_0}\right)^2 + \left(\frac{\partial \Delta p_0^*}{\partial b} b_w\right)^2} \\
    p_{\Delta p_0^*} &= \sqrt{\left(\frac{\partial \Delta p_0^*}{\partial p} p_{\Delta p_0}\right)^2 + \left(\frac{\partial \Delta p_0^*}{\partial p} p_w\right)^2} \\
    u_{\Delta p_0^*} &= \sqrt{b_{\Delta p_0^*} + p_{\Delta p_0^*}}
\end{align*}
\]

The above formulas were applied to three different flow coefficients for each configuration. The final results for the error analysis can be seen on Table 4.4 and Table 4.5 and are also graphically represented on sections 4.3.2 and 4.3.1.

<table>
<thead>
<tr>
<th>(\varphi)</th>
<th>(u_0)</th>
<th>(u_\eta)</th>
<th>(u_\tau^*)</th>
<th>(u_{\Delta p_0^*})</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>0.0372</td>
<td>0.126</td>
<td>6.46E-05</td>
<td>0.00884</td>
</tr>
<tr>
<td>0.2</td>
<td>0.0290</td>
<td>0.0579</td>
<td>0.000162</td>
<td>0.0108</td>
</tr>
<tr>
<td>0.3</td>
<td>0.0352</td>
<td>0.0468</td>
<td>0.000228</td>
<td>0.0189</td>
</tr>
</tbody>
</table>

Table 4.4- Uncertainty results for the “4_19_4” configuration

<table>
<thead>
<tr>
<th>(\varphi)</th>
<th>(u_0)</th>
<th>(u_\eta)</th>
<th>(u_\tau^*)</th>
<th>(u_{\Delta p_0^*})</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.13</td>
<td>0.0172</td>
<td>0.134</td>
<td>0.000114</td>
<td>0.00879</td>
</tr>
<tr>
<td>0.20</td>
<td>0.0407</td>
<td>0.0561</td>
<td>0.000164</td>
<td>0.0180</td>
</tr>
<tr>
<td>0.35</td>
<td>0.0483</td>
<td>0.0310</td>
<td>0.000217</td>
<td>0.0210</td>
</tr>
</tbody>
</table>

Table 4.5- Uncertainty results for the “no guide vanes” configuration

Even though they might not be visible all bar errors are represented for all the flow coefficients and plots. However due to some considerable low uncertainties its representation is not clear which is the case of most of \(T^*\) and \(\Delta p_0^*\).
4.3.3.1 Torque noise filtering

While performing the error analysis one observed that the standard deviation for the recorded values of torque showed very large values. When observing the signal from the torque transducer one realized that it had considerable noise and that was the reason for such a large standard deviation. Moreover the results from the efficiency and torque coefficient plots showed reduced dispersion and they were taken as trust worthy for this reason. However if one considered such standard deviation, the uncertainty would be too large and that would lead to unrealistic intervals of torque related values uncertainty.

To remove the noise from the signal a post-processing manipulation was due. First a list of 3000 samples of torque readings for one specific flow coefficient (\( \phi = 0.2 \)) and rotational velocity (1750 rpm). As the acquisition rate was of 200 Hz this corresponds to a 15 seconds period. Afterwards a “Fast Fourier Transform” was applied to the signal in order to determine which frequencies were to be cut-off and which ones were of interest. Figure 4.11 shows a plot of the transformed signal frequency spectrum.
The point was to attenuate the amplitude of noise with a frequency higher or equal to 25 Hz. To do that a first-order low pass filter was applied to the original signal. First order low pass filter can be represented by the following transfer function:

\[
\frac{\tilde{T}(s)}{T(s)} = \frac{1}{1 + \frac{s}{\tau}}
\]  

(4.18)

Where \( s \) is the Laplace transform variable (frequency domain), \( \tilde{T}(s) \) is the filtered signal, \( T(s) \) is the original signal and \( \tau \) is the time constant which was set to 0.5. That will result in noise reduction and it will return a clearer signal. The result can be seen on Figure 4.12 where one can observe a transient region which was not considered for the new calculation of the standard deviation of the torque.

Comparing Figure 4.10 and Figure 4.12 one can see that noise intensity was substantially attenuated and as a consequence the standard deviation dropped from an absolute value of 3.4 to approximately 0.11 N.m. Error analysis was only performed in a late stage of this work. Given the fact the standard acquisition data routine only returned values of average values and standard deviations, it had to be modified in order to print all the readings over the acquisition data time period. However this was done only after the acquisition data for both configurations was complete. By this time both configurations were disassembled and the new acquisition data method was applied to single rotor with eight blades turbine. After having these results one had to extrapolate for the rest of the experiments. This extrapolation is not abusive since it is believed that the source of noise is the experimental rig structure, more specifically the connections between motor/generator, torque transducer and turbine shaft. These are common to all the configurations analyzed.
4.4 Transverse probe measurements

The transverse probe was used to evaluate total pressures and absolute flow angles on different sections of the two turbine configurations. Some PIV results were also available [14] for comparison for “4_19_4” configuration and its results will also be presented.

4.4.1 “4_19_4” turbine

The plot for the absolute flow angle on section 1 shows that the stator seems to have been well designed given that for radial coordinates away from the hub and casing the flow angle seems to be close to the blade metal angle for $\phi=0.1$ and $\phi=0.2$ with a slight advantage for the latter. The biggest difference comes from the curve respective to $\phi=0.3$ where the flow deflection after the first rotor is as noticeable as for the other two flow coefficients. Common to all the curves are the irregular behavior close to the walls and this is due to the presence of boundary layers that affect the flow angle. Also it should be noted that the transverse probe is not calibrated for near-wall measurements and therefore some errors might arise from this fact. It can also be concluded from the plots for $\phi=0.3$ that flow deflection is clearly affected and one can conclude from this fact the stall already occurred. This flow coefficient is actually un deep stall conditions.

![Figure 4.13 - Absolute flow angle downstream the first rotor for several flow coefficients](image-url)
When it comes to the absolute flow angle downstream the stator, the curves seem to be coincident for

Concerning absolute flow angles downstream the last rotor Figure 4.12, differences are again small regarding the curves for $\phi=0.1$ and $\phi=0.2$ when compared to the one with respect to $\phi=0.3$. According to 2D irrotational flow theory the flow should leave the turbine with no tangential component (for a peak efficiency situation) which only applies in this case if one makes a radial average of the $\phi=0.2$ curve.

4.4.2 “No guide vanes” turbine

The flow deflection downstream the first rotor appears to be identical for $\phi=0.2$ and $\phi=0.13$. It should however be expected that for $\phi=0.2$ the flow deflection would be larger than for $\phi=0.13$. More naturally one can observe that for such a low efficiency point ($\phi=0.35$) the deflection of the flow is of reduced intensity.
When it comes to the flow deflection downstream the second rotor and comparing the same flow coefficient curves, similar comments can be made. The fact that once again curves for $\phi=0.2$ and $\phi=0.13$ are mapped very close to each other might come from the fact that these two flow coefficients do not result in considerable turbine performance differences. In order to notice a more considerable difference one should have picked a much lower flow coefficient. As it happened downstream the first rotor the deflection didn’t prove be as effective for $\phi=0.35$. As a consequence power output is reduced and so is the overall efficiency.

Comparing Figure 4.17 and Figure 4.15 one can see that the guide vanes on the "4_19_4" configuration successfully recover the swirl at turbine outlet when comparing to the "no-guide vanes" turbine. Absolute flow angles at exit rotor outlet show that exit flow has a much larger tangential velocity component when comparing
### 4.5 Integral parameters and loss coefficients

Quantities such as flow rate and torque can be determined using the data acquired by the transverse moving probe. After calculating such parameters the results can be compared with the ones that one gets by means of pressure and torque transducers.

The flow rate can be calculated by the following formula:

\[
Q = \int_0^{2\pi} \int_{r_{\text{hub}}}^{r_{\text{casing}}} \rho \, V \, r \, dr \, d\theta
\]  
(4.19)

The total torque is the sum up of the torque acting on the first rotor, \( T_1 \), and the torque acting on the second rotor, \( T_2 \), and is given by:

\[
T_{\text{aer}} = T_1 + T_2 = \left[ 0 - \int_0^{2\pi} \int_{r_{\text{hub}}}^{r_{\text{casing}}} \rho \, V_{1a}(r) \, V_{1e}(r) \, r^2 \, dr \, d\theta \right] \\
+ \left[ \int_0^{2\pi} \int_{r_{\text{hub}}}^{r_{\text{casing}}} \rho \, V_{2a}(r) \, V_{2e}(r) \, r^2 \, dr \, d\theta - \int_0^{2\pi} \int_{r_{\text{hub}}}^{r_{\text{casing}}} \rho \, V_{3a}(r) \, V_{3e}(r) \, r^2 \, dr \, d\theta \right]
\]  
(4.20)

The latter is based on the assumption that turbine incoming flow as zero tangential velocity component \( V_{r_{\text{in}}} = 0 \).

When it comes to aerodynamic losses three different loss coefficients shall be defined. Several energy (E) balances are performed on the different turbine plane sections from 0 to 4. Regarding the “4_19_4” configuration one can define the following loss coefficients.

The first one is the first rotor loss coefficient:

\[
\zeta_{R1} = \frac{E_0 - E_1 - T_1 w}{Qp_4}
\]  
(4.21)

Where,

\[
E_0 - E_1 = 2\pi \int_{r_{\text{hub}}}^{r_{\text{casing}}} V_{1x} \, r \, (p_{00} - p_{01}) \, dr
\]  
(4.22)

\( p_{00} \) and \( p_{01} \) are the stagnation pressure upstream and downstream the first rotor.

The second loss coefficient is relative to the losses induced by the stator:

\[
\zeta_s = \frac{E_1 - E_2}{Qp_4}
\]  
(4.23)

Where,

\[
E_1 - E_2 = (E_0 - E_3) - (E_0 - E_1)
\]  
(4.24)

and,

\[
E_0 - E_2 = 2\pi \int_{r_{\text{hub}}}^{r_{\text{casing}}} V_{2x} \, r \, (p_{00} - p_{02}) \, dr
\]  
(4.25)

The last loss coefficient is due to the second rotor:

\[
\zeta_{R2} = \frac{E_2 - E_4 - T_2 w}{Qp_4}
\]  
(4.26)
Where, 
\[ E_2 - E_4 = (E_0 - E_4) - (E_0 - E_2) = Qp_4 - (E_0 - E_2) \] (4.27)

It is important to mention that it was chosen \( E_4 \) instead of \( E_3 \) to simulate a section far downstream of the second rotor therefore including losses on the diffuser that are expected to be of reduced importance.

The aforementioned loss coefficients are calculated relatively to the total fluid power (\( Qp_4 = Qp_{04} \)). Similarly, one can define loss coefficients for the “no guide vanes” configuration.

\[ \zeta_{R1} = \frac{E_0 - E_{1=2} - T_1w}{Qp_4} \] (4.28)

\[ E_0 - E_1 = 2\pi \int_{r_{hub}}^{r_{casing}} V_{1x} r (p_{00} - p_{01}) dr \] (4.29)

The notation and definitions used are the same as the ones used previously on the “4_19_4” loss coefficients.

As a gap exists between the two rotors, sections 1 and 2 defined before have no considerable differences and therefore every flow property is considered equivalent. Hence:

\[ \zeta_{R2} = \frac{E_{1=2} - E_4 - T_2w}{Qp_4} \] (4.30)

Where,
\[ E_{1=2} - E_4 = (E_0 - E_4) - (E_0 - E_2) = Qp_4 - (E_0 - E_2) \] (4.31)

### 4.5.1 “4_19_4” turbine

Observing Table 4.3 one can see that the sum up of loss coefficients allow us to calculate the efficiency and compare the results with the ones that resulted from the efficiency experiments. Both methods seem to be in good agreement. Stator losses are considerably low compared to the ones on the rotors, thus validating the stator design method.

<table>
<thead>
<tr>
<th>( \phi )</th>
<th>( \zeta_{R1} )</th>
<th>( \zeta_J )</th>
<th>( \zeta_{R2} )</th>
<th>( \zeta_{total} )</th>
<th>( \eta \text{ calc} )</th>
<th>( \eta \text{ read} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>0.174</td>
<td>0.040</td>
<td>0.212</td>
<td>0.425</td>
<td>0.575</td>
<td>0.544</td>
</tr>
<tr>
<td>0.2</td>
<td>0.195</td>
<td>0.054</td>
<td>0.131</td>
<td>0.380</td>
<td>0.620</td>
<td>0.629</td>
</tr>
<tr>
<td>0.3</td>
<td>0.351</td>
<td>0.046</td>
<td>0.393</td>
<td>0.770</td>
<td>0.230</td>
<td>0.223</td>
</tr>
</tbody>
</table>

Table 4.6- Loss coefficients over different stages of the turbine for three different flow coefficients

Even though calculated efficiencies matched the ones from the efficiency test runs, when it comes to integral parameters such as flow rate and torque larger errors occurred. One can observe that calculated flow rates get more accurate when measured downstream the stator. This is where the resultant value is averaged over the five different tangential positions over which transverse probe measurements were operated. Downstream both rotors the differences cannot be disregarded and can be justified by the time-dependent flow that results from the rotor moving blades.
Moreover the torque calculated by means of integrations schemes don’t match closely with the ones got from the torque transducer. However it should noted that the efficiency was calculated based on calculated values of flow rate and torque.

Table 4.7 - Comparison between data read from efficiency test and calculated data using integration schemes for the “4_19_4” turbine.

<table>
<thead>
<tr>
<th>φ</th>
<th>Qread (m$^3$/s)</th>
<th>Qcalc (m$^3$/s)</th>
<th>Tread (N.m)</th>
<th>Tcalc (N.m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>1.23</td>
<td>1.21</td>
<td>2.68</td>
<td>3.56</td>
</tr>
<tr>
<td>0.2</td>
<td>1.67</td>
<td>1.54</td>
<td>6.41</td>
<td>6.04</td>
</tr>
<tr>
<td>0.3</td>
<td>2.05</td>
<td>1.87</td>
<td>3.57</td>
<td>2.84</td>
</tr>
</tbody>
</table>

“Q read”, “T read” and “η read” stand for flow rate, torque and efficiency read by the efficiency acquisition data routine. Qcalc, Tcalc and η read stand for flow rate, torque and efficiency calculated by means of integration schemes using the transverse probe experiments data.

4.5.2 “No guide vanes” turbine

The calculated efficiency results match with acceptable similarity the ones from the efficiency tests for the three flow coefficients present in Table 4.5. One can observe also that for the three flow coefficients the second rotor is the one responsible for major losses. Comparing peak efficiency flow coefficients for the “4_19_4” and “no-guide vanes turbine” one can see that losses on the second rotor are considerably less significant therefore justifying the presence of an intermediate stator.

Table 4.8 - Loss coefficients over different stages of the turbine for three different flow coefficients

<table>
<thead>
<tr>
<th>φ</th>
<th>ζ rotor1</th>
<th>ζ rotor 2</th>
<th>ζ total</th>
<th>η calc</th>
<th>η read</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.13</td>
<td>0.075</td>
<td>0.478</td>
<td>0.553</td>
<td>0.447</td>
<td>0.501</td>
</tr>
<tr>
<td>0.2</td>
<td>0.144</td>
<td>0.297</td>
<td>0.441</td>
<td>0.559</td>
<td>0.544</td>
</tr>
<tr>
<td>0.35</td>
<td>0.341</td>
<td>0.411</td>
<td>0.751</td>
<td>0.249</td>
<td>0.234</td>
</tr>
</tbody>
</table>

Analyzing calculated flow rates one can see that results show some errors for all the three flow coefficients. The same applies for the torque, being the difference more noticeable for φ=0.35.

Once again the calculated efficiency was derived by making use of the calculated flow rate and torque.
<table>
<thead>
<tr>
<th></th>
<th>Section 1=2</th>
<th>Section 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi = 0.13$</td>
<td>Qread</td>
<td>1.15</td>
</tr>
<tr>
<td></td>
<td>Qcalc</td>
<td>0.97</td>
</tr>
<tr>
<td></td>
<td>Tread</td>
<td>2.34</td>
</tr>
<tr>
<td></td>
<td>Tcalc</td>
<td>2.41</td>
</tr>
<tr>
<td>$\phi = 0.2$</td>
<td>Qread</td>
<td>1.65</td>
</tr>
<tr>
<td></td>
<td>Qcalc</td>
<td>1.55</td>
</tr>
<tr>
<td></td>
<td>Tread</td>
<td>5.84</td>
</tr>
<tr>
<td></td>
<td>Tcalc</td>
<td>5.66</td>
</tr>
<tr>
<td>$\phi = 0.35$</td>
<td>Qread</td>
<td>2.34</td>
</tr>
<tr>
<td></td>
<td>Qcalc</td>
<td>1.98</td>
</tr>
<tr>
<td></td>
<td>Tread</td>
<td>4.04</td>
</tr>
<tr>
<td></td>
<td>Tcalc</td>
<td>4.81</td>
</tr>
</tbody>
</table>

Table 4.9- Comparison between data read from efficiency test and calculated data using integration schemes for the “no guide vanes” turbine.
5 Numerical Analysis

Besides the experimental results presented in the previous chapter this work also includes a numerical analysis that was achieved by means of two commercial software codes. Namely Fluent 6.3 and Gambit 2.3 were used as a solver and mesh generator respectively. CFD codes can be of a lot of help nowadays due to increasing computer calculation capabilities. They first appeared in the sixty’s for aerospace applications and it has gained popularity over the decades. It has been used in early design stages as a reference as well as a comparison to other calculation/design methods. Its use can also avoid the construction of prototypes in early design stages and even though it is expensive to purchase such software it pays-off over the years. The advantage here is not only related to monetary issues but it also has to do with the time effectiveness that CFD codes can return results. The industry adopted this as a needful tool whether commercial or self-developed codes are due. Also a lot of research is being done on new computational codes and this field is still expanding its horizon.

The numerical analysis was performed for the “4_19_4” and “no guide vanes” configurations. For both cases the flow was treated as being incompressible and only a steady-state approach was addressed.

Both geometries were mostly analyzed considering no tip gap. However and given the fact that this approximation might lead to differences between experimental and numerical results a brief tip gap analysis was performed only for the no-guide vanes configuration.

5.1 Governing equations and code

The idea behind this numerical analysis is to perform a 3D analysis of the flow field along each stage of the turbine. To do so the control volume is divided into smaller control volumes called elements (meshing). The method used is the Finite Volume Method. The differential governing equations are discretized and solved by integrating them over each cell. Each variable of interest is located at the centroid of each small control volume. Since fluxes along each face of the cells are conserved, quantities such as mass and momentum in the entire domain are conserved as well. The resultant system of equations is then solved for the entire domain with proper boundary conditions previously imposed.

Since no compressible analysis was addressed the temperature field wasn’t solved (otherwise that would a coupling between energy, continuity and momentum equations), and therefore the energy conservation equation wasn’t a part of this problem. Also no potential flow analysis was regarded leading us to consider only two main equations: continuity equation and momentum conservation equation. However these two equations come in a different form when it comes to consider viscous effects. Since viscous effects can lead to turbulence, and turbulence is certainly present in this case it is important to adapt the governing equations by defining a random property $\varphi$:

$$\varphi(x, t) = \bar{\varphi}(x) + \varphi'(x, t)$$  \hspace{1cm} (5.1)
In the above equation \( \bar{g} \) is the time averaged value of the property and \( g' \) is its fluctuation term. The result is the specific value of that property at a certain time and place.

Making use of the above equation, the continuity equation for a steady-state turbulent flow comes like:

\[
\nabla \cdot \vec{v} = 0
\]

(5.2)

Leading to,

\[
\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0
\]

(5.3)

Where \( u, v \) and \( w \) correspond to the velocity components in the \( x, y \) and \( z \) directions respectively.

If we use now 5.1 in terms of the velocity vector and apply it to the momentum equation it yields to:

\[
\frac{u_i}{\partial x_j} \frac{\partial p}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_l}{\partial x_l} \right) \right] + \frac{1}{\rho} \frac{\partial R_{ij}}{\partial x_i}; \quad i, j = 1, 2, 3
\]

(5.4)

Equations 5.3 and 5.4 are the so-called Reynols-averaged Navier-Stokes or simply RANS equations. They are very similar to the instantaneous form of each equation with exception of the momentum equation that features an extra term accounting for the turbulence effects. Equation 5.4 is written in tensor form and if one further expands it, it would result in three equations one for each direction.

\( R_{ij} \) corresponds to Reynolds tensor (turbulence term) and can be calculated as follows:

\[
R_{ij} = -\rho \bar{u}_i \bar{u}_j
\]

(5.5)

In order to relate the latter with the mean velocity gradients one makes use of the Boussinesq hypothesis:

\[
R_{ij} = \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial \bar{u}_l}{\partial x_l} \right) \delta_{ij}
\]

(5.6)

These are the equations that define a steady-state turbulent flow. However a relation for both turbulent viscosity, \( \mu_t \), and turbulent kinetic energy, \( k \), is still needed in order to solve the system of equations.

Several turbulence models can be chosen and those that can make use of the Boussinesq approach.

### 5.1.1 Standard \( k - \varepsilon \) model

The one used was the \( k-\varepsilon \). This is a two equation model and relates \( \mu_t \) with \( k \) and the turbulent dissipation rate, \( \varepsilon \), as follows:
\[ \rho \frac{\partial}{\partial x_i} (k u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k - \rho \varepsilon \]  \hfill (5.7)

\[ \rho \frac{\partial}{\partial x_i} (\varepsilon u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \]  \hfill (5.8)

\( C_{1\varepsilon}, C_{2\varepsilon}, \sigma_\varepsilon, \sigma_k \) are the turbulence model constants and respectively equal to 1.44, 1.92, 1.3, 1.0.

Where \( G_k \) stands for turbulent kinetic energy and is given by:

\[ G_k = -\rho \overline{u_i u_i} \frac{\partial u_i}{\partial x_i} \]  \hfill (5.9)

This can be related to the Boussinesq approximation by:

\[ G_k = \mu_t S^2 \]  \hfill (5.10)

where \( S \) is the mean rate of strain tensor and is given by:

\[ S \equiv \sqrt{2} S_{ij} S_{ij} \]  \hfill (5.11)

where,

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) \]  \hfill (5.12)

And finally a relation between the turbulent viscosity and \( k \) and \( \varepsilon \) is found:

\[ \mu_t = \frac{\rho C_\mu k^2}{\varepsilon} \]  \hfill (5.13)

\( C_\mu \) is a constant and equal to 0.09.

### 5.1.2 Realizable \( k - \varepsilon \) model

Fluent offers the possibility of choosing the realizable \( k - \varepsilon \) turbulence model. This model better represents flows involving rotation and secondary flow phenomena such as separation and recirculation. One of differences to the standard \( k - \varepsilon \) model is the transport equation for \( \varepsilon \):

\[ \rho \frac{\partial}{\partial x_i} (\varepsilon u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k \sqrt{\varepsilon \nu}} \]  \hfill (5.14)

The other difference concerns the calculation of the turbulent viscosity which is still given by (5.13) however \( C_\mu \) is no longer constant and can be calculated by:
\[ C_\mu = \frac{1}{A_0 + A_2 \frac{k_i}{\epsilon}} \]  
(5.15)

Where,

\[ U^* = \sqrt{S_{ij}S_{ij} + \bar{\Omega}_{ij}\bar{\Omega}_{ij}} \]  
(5.16)

\[ \bar{\Omega}_{ij} = \Omega_{ij} - 2\epsilon_{ijk}w_k \]  
(5.17)

\[ \Omega_{ij} = \bar{\Omega}_{ij} - \epsilon_{ijk}w_k \]  
(5.18)

\[ A_2 = \sqrt{6} \cos \phi \]  
(5.19)

\[ \phi = \frac{1}{3} \cos^{-1}(\sqrt{6}W) \]  
(5.20)

\[ W = \frac{S_{ij}S_{jk}S_{ki}}{S^3} \]  
(5.21)

\[ S = \sqrt{S_{ij}S_{ij}} \]  
(5.22)

### 5.1.3 Discretization and other computational parameters

The previous turbulence schemes are not able to solve the governing equations in the near-wall region more specifically in the viscous sub-layer where typically the non-dimensional parameter \( y^* < 5 \). To overcome this limitation a wall function is employed. The wall function makes the bridge between the fully turbulent outer region and viscous affected inner region. This avoids the employment of a very fine mesh close to the wall making it less expensive from a computational point of view as well as easier to mesh as it was in case of the stator volume. However in order to reasonably solve the wall function the value of \( y^+ \) has to be comprehended between 30 and 300.

The ambient pressure was set to 101325 Pa for all simulations. Regarding air properties viscosity, \( \mu \), was 1.789x10^{-5} Pa.s and air density, \( \rho \), featured a constant value of 1.225 kg/m³.

The numerical model used by this software solves the equations of continuity and momentum using the segregated model. This means that equations are solved in a sequential way. The coupling between velocity and pressure was achieved by means of SIMPLEC algorithm which usually obtains a faster convergence of the solution, according to [12].

All the simulations run on a simple precision accuracy and equations and its variables were discretized with second order schemes as shown in Table 5.1:
Discretization

<table>
<thead>
<tr>
<th>Equation</th>
<th>Discretization Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>Second Order</td>
</tr>
<tr>
<td>Momentum</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Turbulent Dissipation Rate</td>
<td>Second Order Upwind</td>
</tr>
</tbody>
</table>

Table 5.1- Equation discretization types

5.2 Moving reference frame formulation

The CFD analysis featured two rotors and one stator. This means that calculations on both rotating and static surfaces are due. In order to better approach this problem two reference frames were set: one static and one other that was rotating with the rotor at the same angular velocity. The control volumes that include the first and second rotor are solved in a non-inertial reference frame and the stator control volume is solved in an inertial reference frame. The reason for choosing a moving reference frame is because by doing so the rotor flow can be solved as being steady over the rotor blades. When one sets a moving reference frame one is inducing a velocity to the mesh cells of the rotating domain. Since rotation origins acceleration new terms in the governing equations will rise up. Fluent allows the user to choose if the velocity formulation is done on a relative or absolute basis.

The relation between absolute velocity, \( v_i \), and relative velocity, \( w_i \):

\[
 w_i = v_i - e_{ijk} \Omega_j \times r_k \tag{5.23}
\]

It should be noted that \( w \) only stands for relative velocity for this particular sub-section and should not be confused with the absolute velocity vector in the \( z \) direction as it is in the rest of this text.

Where \( \Omega_j \) is the angular velocity of the rotating frame and hence the angular velocity of the rotors and \( r_k \) stands for location vector of the non-inertial frame. Applying the above equation to the momentum equation it comes that:

\[
 w_j \frac{\partial w_i}{\partial x_j} + e_{ijk} \Omega_j w_k + e_{ijk} \Omega_j e_{ktn} \Omega_t r_m = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial w_i}{\partial x_j} \right] \tag{5.24}
\]

Where \( e_{ijk} \Omega_j w_k \) represents the Coriolis acting on the fluid due to rotation. In terms of relative velocities the continuity equation comes simply as follows:

\[
 \frac{\partial w_i}{\partial x_j} = 0 \tag{5.25}
\]

Both reference frames have its origin at the same point and the axial directions are coincident with the geometrical axis of the turbine. The simulations were made at constant rotor angular velocity and therefore the rotation speed of the moving reference frame was set to 2000 rpm.
5.3 Control volume and mesh

The control volumes regarding both geometries that were analyzed were designed by means of commercial CAD software: Solidworks. CAD software allows easy and quick manipulation of geometric entities and the resultant files can be saved in a format (*.sat) that can be imported afterwards to the mesh generator software used: Gambit 2.3.

5.3.1 “4_19_4” turbine

The control volumes didn’t include a complete geometrical reproduction of the turbines. The flow inside the turbine is considered to be tangentially periodic and therefore only a fraction of the annular cross-section of the turbine was reproduced. By making this assumption it is assumed that the flow pattern on the remaining fractions of the turbine is equal to the reproduced one and hence their representation can be dismissed. The annular section fraction is determined by the number of blades of each section. This implies that on the case of the rotor domains where 4 blades are mounted the annular cross-section that is represented is divided by 4. So the periodic boundaries of the domain are 90° distant from each other. In case of the stator that features 19 blades the simulation domain is only 1/19 of the real one resulting in periodic boundaries approximately 18.9° away from each other. The convergent nozzle at turbine inlet wasn’t considered for the simulations as well as the tip gap for the sake of simplicity. The inlet of the computational domain was located at two rotor blade chords upstream from the first rotor so that no remarkable influence on the rotor could occur during the calculations. On the other hand the outlet was seven rotor blade chords downstream the second rotor.

By doing this it is expected the outlet flow is far away from the influence of the second rotor.

The stator domain was created by first stacking several cross-sectional curves of the vane. The thickness of the blade was the same as the real one being 2mm and the leading and trailing edges featured round edges of 1mm if radius. After the blade has been created two periodic surfaces tangentially

Figure 5.1- "4_19_4" computational domain (casing and periodic surfaces not represented).
equispaced from the guiding vane were created. Finally an extension of both periodic boundaries was set so that real stator hub dimensions could be respected. The dimensions regarding the relative distance between stator and rotors were kept as in the real model as well as hub and casing diameters.

The mesh used on both rotors domains was a structured mesh. A structured mesh has some benefits in the way that mesh quality and density can be better predicted and controlled. A C-type mesh was applied to the regions near the rotor blades as illustrated in Figure 5.2a). Such a mesh has proved to be reliable in Wells turbine CFD analysis [13] [14]. Only hexahedral elements were applied for such domains. Attention was focused on the regions where larger gradients were likely to be present, for example in regions close to solid walls as it is the case of blades. More specifically a finer mesh was applied near the leading and trailing edges as well as the region of the blade where stagnation point is expected to be. Also a finer mesh was created near the hub and casing becoming coarser as one reaches the mid-span coordinate. No boundary layer was created around the blades since the height of the first elements could be easily controlled and the blade adjacent mesh was itself of the type of a boundary layer having only quadrilateral elements.

The stator mesh geometry is fully structured and consisted of hexahedral elements only Figure 5.3. A boundary layer mesh was created for the hub and casing surfaces with 2.5mm height for both cases. However a good indicator if whether the first height cells are correctly set is the value of the non-dimensional parameter $y^+$. Plots of $y^+$ can be analyzed ahead in chapter 6. Higher concentration of elements was again directed towards the leading and trailing edges and mesh grew finer towards casing and hub in the radial direction. Similarly finest mesh was located near guide vane leading and trailing edges. Three different grids were created G1S, G2S and G3S. The point is to try to evaluate the differences that finer meshes can have on the computational results. Meshes with a larger number of elements are more expensive form a computational point of view and sometimes a higher degree of refinement doesn’t lead to any worthwhile improvement on the final results. A summary of the main mesh characteristics is present in Table 5.2:
The amount of elements that are present in a certain mesh isn’t the only parameter that should be taken into account. The quality of a mesh is also important so that the integrating process during the calculations can be done with smaller errors. Among others an important parameter that defines the quality of a mesh is the angles between each edges/faces of an element. These angles should be as close as possible to the ones that feature an element with equal edge length. The equi-angle skew is the parameter that allows us to evaluate this property. It is defined as follows:

![Figure 5.3- Stator hub and blade mesh](image)

<table>
<thead>
<tr>
<th>Grid</th>
<th>Domain</th>
<th>Tangential</th>
<th>Axial</th>
<th>Radial</th>
<th>Blade</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>G3S</td>
<td>Rotor 1</td>
<td>120</td>
<td>95</td>
<td>25</td>
<td>60</td>
<td>881000</td>
</tr>
<tr>
<td></td>
<td>Stator</td>
<td>40</td>
<td>100</td>
<td>25</td>
<td>80</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Rotor 2</td>
<td>120</td>
<td>155</td>
<td>25</td>
<td>60</td>
<td></td>
</tr>
<tr>
<td>G2S</td>
<td>Rotor 1</td>
<td>140</td>
<td>95</td>
<td>30</td>
<td>80</td>
<td>1314000</td>
</tr>
<tr>
<td></td>
<td>Stator</td>
<td>50</td>
<td>160</td>
<td>30</td>
<td>120</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Rotor 2</td>
<td>140</td>
<td>155</td>
<td>30</td>
<td>80</td>
<td></td>
</tr>
<tr>
<td>G1S</td>
<td>Rotor 1</td>
<td>180</td>
<td>95</td>
<td>30</td>
<td>120</td>
<td>1797000</td>
</tr>
<tr>
<td></td>
<td>Stator</td>
<td>60</td>
<td>180</td>
<td>30</td>
<td>160</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Rotor 2</td>
<td>180</td>
<td>155</td>
<td>30</td>
<td>120</td>
<td></td>
</tr>
</tbody>
</table>

Table 5.2-Element distribution and total number of elements for the 4_19_4 turbine model
Where $\theta_{\text{max}}$ is the maximum angle between edges, $\theta_{\text{min}}$ is the equivalent minimum and $\theta_{\text{eq}}$ is the angle between edges on a equi-sized edges element. For a hexahedral element $\theta_{\text{eq}} = 90^\circ$ and for a wedge element $\theta_{\text{eq}} = 60^\circ$. This normalized parameter should be close as close to zero as possible hence indicating a low distortion of elements.

A summary on the elements skewness of the three meshes analyzed is presented on table 5.3.

$$Q_{EAS} = \max \left\{ \frac{\theta_{\text{max}} - \theta_{\text{eq}}}{180 - \theta_{\text{eq}}}, \frac{\theta_{\text{eq}} - \theta_{\text{min}}}{\theta_{\text{eq}}} \right\}; \quad 0 \leq Q_{EAS} \leq 1 \quad (5.26)$$

### 5.3.2 “No guide vanes” turbine

The meshing parameters of the rotor blades on the “no guide vanes” turbine were the same as the ones applied in G2 mesh. The 3D mesh is fully structured. The gap between rotors has been meshed with hexahedral elements. Also in terms of tangential direction the number of divisions still matches the aforementioned mesh. Tables 5.4 and 5.5 resume the mesh parameters of the grid created for this configuration. This mesh will be called H2.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Domain</th>
<th>Tangential</th>
<th>Axial</th>
<th>Radial</th>
<th>Blade</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>H3</td>
<td>Rotor 1</td>
<td>120</td>
<td>130</td>
<td>25</td>
<td>60</td>
<td>940 000</td>
</tr>
<tr>
<td></td>
<td>Rotor 2</td>
<td>120</td>
<td>170</td>
<td>25</td>
<td>60</td>
<td>940 000</td>
</tr>
<tr>
<td>H2</td>
<td>Rotor 1</td>
<td>140</td>
<td>130</td>
<td>30</td>
<td>80</td>
<td>1 308 000</td>
</tr>
<tr>
<td></td>
<td>Rotor 2</td>
<td>140</td>
<td>170</td>
<td>30</td>
<td>80</td>
<td>1 308 000</td>
</tr>
<tr>
<td>H1</td>
<td>Rotor 1</td>
<td>180</td>
<td>130</td>
<td>30</td>
<td>120</td>
<td>1 668 000</td>
</tr>
<tr>
<td></td>
<td>Rotor 2</td>
<td>180</td>
<td>170</td>
<td>30</td>
<td>120</td>
<td>1 668 000</td>
</tr>
</tbody>
</table>

Table 5.4- Element distribution for the “no guide vanes” configuration
5.3.2.1 “No guide vanes” turbine tip gap analysis

The radial distance between blade tip and casing was set to 1 mm as it would be close to the experimental rig reality. Tip gap mesh was executed according to what has been done by [17]. The volume has been radially divided into two ensuring $30<y^+<300$ condition on both casing and blade tip could be met. Mesh is radially structured and the volume mesh consists only on wedge elements.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Skew angle</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0-0.25</td>
</tr>
<tr>
<td>H3</td>
<td>97.07%</td>
</tr>
<tr>
<td>H2</td>
<td>91.62%</td>
</tr>
<tr>
<td>H1</td>
<td>95.21%</td>
</tr>
</tbody>
</table>

Table 5.5- Element skewness ratio in terms of total number of elements for the “no guide vanes” configuration

5.4 Boundary conditions

Boundary conditions can have a big effect on computational simulations. A correct set of boundary conditions is very important to get a trust-worthy solution. Several boundary conditions may sometimes apply for the same geometrical entity. The task is therefore to choose the ones that better simulate reality.

Figure 5.4- Blade tip mesh detail on leading and trailing edges
5.4.1 Turbine with 19 bladed stator

5.4.1.1 Inlet
Pressure inlet boundary conditions were applied at the turbine inlet surface. In detail a total pressure of 0 Pa was set at all inlets. This is supposed to simulate a far upstream area where a fluid particle is far away from the turbine inlet.

Additional parameters used to define this velocity inlet boundary condition: hydraulic diameter and turbulent intensity. According to [12] these are suitable inputs when one has flows that go through rotating vanes. Turbulent intensity was assumed to be low and therefore a value of 1% was set for this parameter since the flow upstream is undisturbed. Hydraulic diameter is the parameter acting as the characteristic eddy length scale. The hydraulic diameter can be calculated by the following expression:

\[ D_h = \frac{4A}{P} \]  

where \( A \) (m²) is the cross-sectional area and \( P \) (m) is the wetted perimeter. For the annular duct it simply results in:

\[ D_h = D_{ext} - D_{int} = 0.590 - 0.400 = 0.190 \text{ m} \]

5.4.1.2 Solid Wall
Solid wall boundary conditions are due whenever we are in the presence of material surfaces such as blades, casing or hub. However some differences might be present while setting these boundary conditions as some surfaces are rotating. The rotating physic walls in the computational model include rotor blades as well as the portion of the rotor that bears the blades that is itself part of the hub. These surfaces are rotating in the static reference frame. However as described before the rotor domains have a rotating reference frame and hence the latter is set to be static with respect to this frame. The remaining solid walls that are part of the rotor domains are set as be rotating at 2000 rpm along with moving reference frame. By doing this the relative velocities between moving and static parts can be maintained. Solid walls are boundary conditions that impose zero normal velocity across the surfaces where it is applied. Being a viscous flow the no-slip condition was also imposed meaning that tangential velocities are set to zero along the walls. The roughness constant was left at the default value of 0.5 and roughness height of 0 as it is assumed that walls have constant roughness and also a smooth surface.

5.4.1.3 Mixing Plane
When one has domains with relative velocities an interface between them is due. There are two rotating domains and one static domain in between. As a consequence two interfaces have to be
present in each end of the stator. Being a steady-state analysis the interface is achieved by setting up a mixing plane. Since rotor and stator have different number of blades the volumes do not have the same tangential amplitude and therefore the mixing plane has to average every flow property from one volume to the other. For each iteration the variable values of the upstream side of the mixing plane are spatially averaged and then updated to the downstream side of the plane and this process goes on until convergence is achieved.

The mixing plane is one of constant axial coordinate. The average is done spatially over the tangential direction. This means that all non-uniformities coming for example from wakes and separation disappear when flow properties are passed to the new domain. Therefore flow properties will only vary radially on a mixing plane. Several boundary condition pair can be applied upstream and downstream of the mixing plane. The ones chosen are the same on both mixing planes and can be seen on table

<table>
<thead>
<tr>
<th>Mixing Planes</th>
<th>Upstream</th>
<th>Downstream</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor 1 Outlet- Pressure outlet</td>
<td></td>
<td>Stator Inlet- Pressure inlet</td>
</tr>
<tr>
<td>Stator Outlet- Pressure outlet</td>
<td>Rotor 2 Inlet- Pressure inlet</td>
<td></td>
</tr>
</tbody>
</table>

Table 5.6- Mixing plane boundary conditions

When one imposes pressure boundary conditions on both upstream and downstream sides of a mixing plane mass conservation is not assured. Such a fact can be outlined if one imposes a mass flow condition on the downstream side of the plane. However a bigger importance was attributed to the pressure field across the several domains hence the choice of such boundary conditions.

Each of the mixing planes was radially divided into the same number of divisions of the mesh at each case. The number of interpolation points dictates the number circumferential averages along the radial direction.
5.4.1.4 Periodic Boundaries

Periodic boundaries include all the lateral bounding surfaces of the total computational domain. The periodic boundaries present in this case are of the rotational type. The setting up of such boundary conditions implies that the flow field on one of the surfaces matches the one on the opposite side. Periodic boundaries can therefore account for the presence of the neighboring blades even though there are not represented.

5.4.1.5 Outlet

A pressure outlet condition was imposed at the outlet of the domain. More specifically values varying from -900 to -1700 Pa imposed to the outlet surface. Since a constant total pressure was set at the inlet one has to vary the static pressure at turbine outlet in order to simulate several flow coefficient scenarios. These values are relative to the operating pressure (101325 Pa). At the outlet surface the radial equilibrium condition was imposed. This means that the value of static pressure previously set applies only for the hub line and static pressure will vary according to:

\[
\frac{\partial p}{\partial r} = \frac{\rho V_\theta^2}{r} \tag{5.29}
\]

Where \( r \) is the distance to the axis of rotation and \( V_\theta \) is the tangential velocity. The latter is based on the assumption that no remarkable radial component of velocity is present. This is a fair assumption since it is an axial flow machine.

5.4.2 Turbine with no guide vanes

The simulations performed on the “no guide vanes” configuration were slightly different from the ones applied for the configuration with the stator in between the two rotors. One important difference is the fact that no mixing planes were needed for this turbine. The reason for that is because the geometry maintains its tangential amplitude all along the axial direction, from inlet to outlet, as shown in Figure 5.8. Furthermore only one reference frame (rotational) was applied since the stator (non-rotating) domain is no longer part of the geometry. For the rest of the parameters such as boundary conditions, discretization and convergence criteria no alteration took place.
5.5 Numerical uncertainty

There are three different factors that contribute to numerical errors. The first one is the round-off error that arises from the fact that computers have limited precision during the calculation process. This error becomes more important with increasing mesh refinement. It should be noted that all the simulations were run with single precision variables.

Iterative errors come from the non-linearity of the system of equations that are transformed into a linear system of equations.

In order to calculate numerical uncertainty one assumes that round-off and iterative errors can be neglected when compared to the discretization error.

Discretization errors are closely related to the degree of mesh refinement. It can be pointless to further refine a certain mesh if it doesn’t pay off in terms of result accuracy. Furthermore the finer the mesh the more expensive it becomes to solve from a computational point of view. This may lead to unnecessary computational time spent during the process.

The uncertainty of a certain flow property can be calculated by Richardson extrapolation method:

\[ e_d(\phi) = \phi_i - \phi_0 = a(h)^p \]  

(5.30)

Where \( \phi_i \) is the solution of a particular mesh, \( \phi_0 \) is an estimate of the real solution, \( a \) is a constant and \( p \) is the apparent order of convergence. \( h \) is a parameter that indicates the degree of refinement of the mesh and can be calculated by:

\[ h = \sqrt[3]{\frac{Vol}{N_{elem}}} \]  

(5.31)
When calculating uncertainty (U) one should observe that the calculated property (\(\phi\)) should fall between the interval defined by \([\phi_0 - U, \phi_0 + U]\).

Uncertainty can be calculated by the Grid Convergence Index as follows:

\[
U = F_s |e_d| \tag{5.32}
\]

Where \(F_s\) is a constant that varies with the type of convergence [18]. However whenever one doesn’t get monotonic convergence the uncertainty can be calculated using the maximum difference between the results regarding different mesh simulations [19]:

\[
U = 3\Delta M \tag{5.33}
\]

Given the complexity of the simulations performed noise will be most certainly present and therefore one chose the last method.

In order to calculate uncertainty three flow properties were chosen based on its importance throughout this work. These are torque, total pressure at turbine outlet and mass flow rate.
6 Numerical results

The numerical results to be compared with the experimental ones are relative to meshes G2 and H2. The other meshes mentioned on Chapter 5.3 will however be addressed to judge the numerical error of the simulations.

6.1 Solution convergence

The convergence of all the simulations was judged based on different parameters. The first one was setting a minimum value of $10^{-5}$ for all residuals. All residual ended up between $1*10^{-6}$ and $1*10^{-7}$ except for the continuity one that stabilized around $10^{-5}$. Moreover convergence was considered to be achieved if other integral computational parameters stabilized around constant values. This happened always before the residual values reached the target values. Velocity magnitude was the quantity analyzed at both computational domains inlets, whereas velocity magnitude was the parameter to be followed at outlets.

6.2 y+ verification

In order to properly solve the governing equations near the wall a suitable first cell height close to solid walls must be set. Every simulation performed resulted in a range of $y^*$ mostly between the desired values ($30<y^*>300$). Next it will be presented the values for one flow coefficient of each configuration analyzed.

![Graph](image)

Figure 6.1 - $Y^+$ values for one flow coefficient of each configuration. "No guide vanes" on the left hand-side and "4_19_4" on the right hand-side

6.3 Integral parameters and comparison with experimental results

6.3.1 “4_19_4” turbine

When using mixing planes it is important to check for property unbalance across mixing planes. Mass flow unbalance across mixing planes was however expected due to limitations of the numerical code. Since pressure inlet and pressure outlet boundary conditions were set the property that will
remain constant across mixing planes will be the pressure. This can be confirmed when one looks at very low pressure unbalance.

Axial velocity is responsible for mass flow conservation and one can see that this is the field where larger differences take place across tangential averages. These differences were however considered not to have great impact on final results.

<table>
<thead>
<tr>
<th></th>
<th>Upstream mixing plane</th>
<th>Downstream mixing plane</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Rotor 1 outlet</td>
<td>Stat or inlet</td>
</tr>
<tr>
<td>Pressure (Pa)</td>
<td>-413.25</td>
<td>-413.22</td>
</tr>
<tr>
<td></td>
<td>-0.10%</td>
<td>-0.10%</td>
</tr>
<tr>
<td>Axial Velocity (m/s)</td>
<td>6.96</td>
<td>7.00</td>
</tr>
<tr>
<td></td>
<td>+0.57%</td>
<td>+1.70%</td>
</tr>
<tr>
<td>Flow angle (deg)</td>
<td>-40.81</td>
<td>-39.83</td>
</tr>
<tr>
<td></td>
<td>-2.46%</td>
<td>+2.74%</td>
</tr>
</tbody>
</table>

|                   | Pressure (Pa)         | Axial Velocity (m/s)   | Flow angle (deg)       |
|                   | -605.14               | 10.07                  | -41.06                 |
|                   | -0.12%                | -1.31%                 | 0.76%                  |
|                   | -505.07               | 9.94                   | -40.75                 |
|                   | -0.12%                | -1.31%                 | 0.76%                  |
|                   | -618.33               | 9.99                   | 31.35                  |
|                   | -0.10%                | +0.5%                  | 32.02                  |
|                   | -618.27               | 10.04                  |                        |
|                   |                       |                        |                        |
|                   | -929.06               | 16.98                  | -39.60                 |
|                   | 0.00%                 | -0.18%                 | -0.25%                 |
|                   | -929.06               | 16.95                  | 39.59                  |
|                   | 0.00%                 | -0.18%                 | +0.53%                 |
|                   | -960.56               | 16.97                  | 32.14                  |
|                   | -0.16%                | +1.36%                 | 32.31                  |
|                   | -960.41               | 17.20                  |                        |

Table 6.1- Properties integration differences across mixing-planes for the structured mesh G2S

Looking at torque coefficient plot one can see that torque is over calculated. This trend has been reported to happen even if tip gap is modeled [17], which hasn’t been the case. Tip gap with result however in a lower torque and energy dissipation due to vortex shading and inducted radial velocities [17]. Stalling has been anticipated by the numerical code. Total pressure drop coefficient fairly reproduced experimental reality, except for the last flow coefficient point. This resulted in an overall efficiency that is somehow larger for most flow coefficients. Peak efficiency is 5% above what has been calculated in the experimental rig.
Figure 6.2- Comparison between experimental efficiency curves and numerical efficiency curve

Figure 6.3- Total pressure drop coefficient comparison between experimental results and numerical results
Similarly to what happened to the “4_19_4” configuration the “no guide vanes” configuration over-calculates torque. This however was expected to happen since no tip gap was considered in the geometry. Tip gap is a geometric feature that plays an important role on the secondary flow phenomenon that characterizes the flow on a Wells turbine. As a consequence the narrower the space between blade tip and casing becomes the higher the efficiency gets. As a consequence of an inexistent tip gap the numerical simulations result in a higher efficiency along the entire flow coefficient range. Predictably the same explanation stands for the torque coefficient analysis. Since there’s no vortex generated in the tip clearance region less kinetic energy is dissipated and torque becomes larger. Stalling occurs for the same flow coefficient in both numerical and experimental results.

Total pressure drop coefficient curves are almost coincident, being the flow property more accurately calculated when compared to the experiments.

![Figure 6.4- Torque coefficient comparison between experimental and numerical results](image)

![Figure 6.5- Efficiency curve comparison between experimental and numerical results](image)
Regarding the absolute flow angle downstream the first rotor one can observe that in line with experiment, the plots regarding $\phi = 0.2$ and $\phi = 0.1$ are very close and in a region half way between hub and casing even coincident. The major differences between empirical and numerical data regard the post stalling flow coefficient, especially in the region close to the hub. Nevertheless it can be noted that the flow deflection for $\phi = 0.37$ is clearly less efficient.

Figure 6.6- Total pressure drop coefficient comparison between experimental and numerical results

Figure 6.7- Torque coefficient comparison between experimental and numerical results
Similar comments can be made when it comes to the absolute flow angle downstream the second rotor. However analyzing plots from $\phi = 0.2$ and $\phi = 0.1$ one can notice in a region close to the casing the differences are noticeable. This could justify the larger torque calculated by the numerical simulations. Again discrepancies can be observed on the plots regarding post-stall conditions.

### 6.3.3 Tip gap effect on simulations

A brief analysis was performed in order to evaluate tip gap effect on turbine efficiency. Wells turbine tip gap effect is known to have significant influence on turbine performance. It was expected to affect negatively the efficiency results. That behavior turned out to be confirmed and for a peak efficiency flow coefficient ($\phi = 0.20$) the following results were obtained:
Although results show better agreement with experimental results a difference of 4% is still achieved.

### 6.4 Error analysis

#### 6.4.1 “4_19_4” configuration

From the results shown in this section it can be seen mesh refinement resulted in considerable differences for the unstructured mesh. Regarding the structured mesh it can be seen that mesh convergence is better achieved. Uncertainty values are much smaller and can thus validate the mesh quality. Results considerably improve when comparisons are made between structured and unstructured mesh.

<table>
<thead>
<tr>
<th>Variable</th>
<th>U</th>
<th>U/\phi [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>T</td>
<td>2.191</td>
<td>0.075</td>
</tr>
<tr>
<td>P(_0)</td>
<td>-1412.27</td>
<td>15.765</td>
</tr>
</tbody>
</table>

#### 6.4.2 “No guide vanes” configuration

This configuration holds small uncertainty values and its fully structured mesh is trust-worthy as no considerable differences occur with increasing mesh refinement.

<table>
<thead>
<tr>
<th>Variable</th>
<th>U</th>
<th>U/\phi [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>T</td>
<td>1.178</td>
<td>1128.101</td>
</tr>
<tr>
<td>P(_0)</td>
<td>1.177</td>
<td>1130.200</td>
</tr>
<tr>
<td>P(_1)</td>
<td>1.175</td>
<td>1130.296</td>
</tr>
<tr>
<td>Variable $\phi$</td>
<td>$U$</td>
<td>$U/\phi[%]$</td>
</tr>
<tr>
<td>---------------</td>
<td>-------</td>
<td>--------------</td>
</tr>
<tr>
<td>$T$</td>
<td>1.175</td>
<td>0.00921</td>
</tr>
<tr>
<td>$P_0$</td>
<td>1130.296</td>
<td>6.585</td>
</tr>
</tbody>
</table>

Table 6.6- Parameter uncertainty for finest mesh
7 Conclusion

Conclusions can be taken at different levels. Let us first look at the comparison between the two different tested turbine configurations: "4_19_4" and "no guide vanes". The former seem to have an overall better performance if one observes the plots for the torque and pressure drop coefficients. Consequently the efficiency curve reaches higher peak efficiency, 64%. Oppositely the "no guide vanes" turbine efficiency only gets to 57%. One can therefore conclude an obvious advantage in the presence of an intermediate stator. Swirl related losses at turbine outlet have been therefore minimized and that had positive impact on the efficiency. Calculated loss coefficients revealed that no major losses took place over the stator domain. One can observe the losses mainly occurred through the rotors, more specifically on the downstream rotor where losses were always larger comparing to the upstream one. The same happened to the "no guide vanes" configuration.

Comparing to the monoplane Wells turbine with guide vanes the overall performance is however still worse. The latter reaches a peak efficiency of 71% and holds as a better option regarding exclusively efficiency criteria, however it also showed a narrower range of flow coefficient usability. The "4_19_4" proved to have worse performance than expected at first. Results were to be close to its term of comparison and the difference is considerable.

Transverse probe measurements revealed some differences in flow rate when measuring sections downstream both rotors and comparing to what was measured in the efficiency tests. This difference was less considerable when evaluating the section downstream the stator. Wake instabilities downstream Wells turbine rotors seem to be the cause of such fact.

"4_19_4" numerical simulations showed slight mass unbalance across mixing planes. Overall efficiency has been higher than experimental results and reached a peak of 68%. Since pressure drop have been in agreement with experiments the efficiency have been essentially affected by torque calculations. Mesh refinement errors showed acceptable values and therefore validate the mesh created.

Looking at the "no guide vanes" numerical simulations one can conclude that due to the fact that no tip clearance was modeled the results regarding torque were increased. Consequently efficiency reached a higher peak in comparison to experimental data. Stalling however was well predicted and pressure drop remained close to what was measured at the rig. The brief one flow coefficient tip gap simulation resulted as expected in a torque and efficiency reduction (4%). Numerical error analysis proved the three meshes with different degrees of refinement to be consistent and therefore trustworthy.
8 Future work

Given that this work only addressed two of the three configurations initially planned one should evaluate the configuration with the six bladed rotors and the twenty-nine blades stator. Its performance should be compared to the results this work presents. Tip gap simulations should be performed also for the “4_19_4” geometry to make a comparison to the experimental results. This would also validate conclusions on tip gap simulation effect on “no guide vanes” simulations. Also as mixing-planes always introduce errors in calculations a sliding mesh approach would be important. To do so it should be considered a 20 blades stator instead of 19. This would avoid a 360° domain simulations and would be considerably less expensive computationally speaking. This approximation would also have to be taken into consideration when analyzing results and comparing with the real model geometry.
Bibliography


[18] P.J. Roache, Verification and Validation in Computational Science and Engineering.: Hermosa


[37] M. Folley et al, "Comparison of LIMPET contra-rotating Wells turbine with theoretical and model


Appendix

A.1 Single plane Wells turbine with guide vanes (same rotor solidity) [8]

Appendix Figure 1- Efficiency and total pressure drop coefficient curves (filled squares and triangles)

Appendix Figure 2- Torque coefficient curve (filled triangles)
A.2 Project technical drawings

Appendix Figure 3- Hub steel plate with guiding vane cuts before roll bending (part 1/2).
Appendix Figure 4 - Hub steel plate with guiding vane cuts before roll bending (part 2/2).

Appendix Figure 5 - Casing steel plate with guiding vane cuts before roll bending (part 1/2).
Appendix Figure 6- Casing steel plate with guiding vane cuts before roll bending (part 2/2).

Appendix Figure 7- Plan stator blade before roll bending.
Appendix Figure 8- Steel structure that bears the bearings and shaft.

Appendix Figure 9- Shaft
Appendix Figure 10- Steel bushing making the union between stator and stell structure that holds the shaft

Appendix Figure 11- Rotor end plate
Appendix Figure 12- Bearing stopper

Appendix Figure 13- Part connecting shaft and rotor hub
Appendix Figure 14- Rotor hub

Appendix Figure 15- Rotor end plate
Appendix Figure 16- Steel structure that holds the turbine model
A.3 Stator data

<table>
<thead>
<tr>
<th>Radius (m)</th>
<th>0,200</th>
<th>0,210</th>
<th>0,219</th>
<th>0,229</th>
<th>0,238</th>
<th>0,248</th>
<th>0,257</th>
<th>0,267</th>
<th>0,276</th>
<th>0,286</th>
<th>0,295</th>
</tr>
</thead>
<tbody>
<tr>
<td>R*</td>
<td>0,00</td>
<td>0,10</td>
<td>0,20</td>
<td>0,30</td>
<td>0,40</td>
<td>0,50</td>
<td>0,60</td>
<td>0,70</td>
<td>0,80</td>
<td>0,90</td>
<td>1,00</td>
</tr>
<tr>
<td>Alfa 2 (°)</td>
<td>45,50</td>
<td>43,67</td>
<td>41,90</td>
<td>40,29</td>
<td>38,73</td>
<td>37,21</td>
<td>35,81</td>
<td>34,50</td>
<td>33,21</td>
<td>31,98</td>
<td>30,90</td>
</tr>
<tr>
<td>Alfa L2 (°)</td>
<td>53,13</td>
<td>51,51</td>
<td>49,94</td>
<td>48,52</td>
<td>47,14</td>
<td>45,79</td>
<td>44,57</td>
<td>43,41</td>
<td>42,27</td>
<td>41,17</td>
<td>40,22</td>
</tr>
<tr>
<td>c/t (·)</td>
<td>263,14</td>
<td>245,80</td>
<td>229,91</td>
<td>215,71</td>
<td>202,63</td>
<td>190,54</td>
<td>179,63</td>
<td>169,64</td>
<td>160,32</td>
<td>151,68</td>
<td>144,01</td>
</tr>
<tr>
<td>Chord (m)</td>
<td>0,17</td>
<td>0,17</td>
<td>0,17</td>
<td>0,16</td>
<td>0,16</td>
<td>0,16</td>
<td>0,15</td>
<td>0,15</td>
<td>0,15</td>
<td>0,14</td>
<td>0,14</td>
</tr>
<tr>
<td>Blade Radius (m)</td>
<td>0,11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>f/c (%)</td>
<td>0,20</td>
<td>0,20</td>
<td>0,19</td>
<td>0,19</td>
<td>0,18</td>
<td>0,18</td>
<td>0,18</td>
<td>0,17</td>
<td>0,17</td>
<td>0,16</td>
<td>0,16</td>
</tr>
</tbody>
</table>

Appendix table 1- stator data

Alfa L1 and Alfa L2 are the blade metal angles. These angles are therefore the ones that dictate the geometrical manufacturing limits of the blades. As stated in the table they are symmetric with respect to the axial direction and this allows the turbine to be able to work in a cyclic oscillating flow pattern environment. This symmetry will assure equivalent flow conditions from whatever side of the turbine the flow is coming from.