Hydropower for sustainable development:

CFD modelling and hydro-mechanical behavior of micro hydro converters for low head pressure flow

Roberta Zimmitti

Thesis to obtain the Master of Science Degree in

Mechanical Engineering

Supervisors: Prof. João Eduardo de Barros Teixeira Borges

Prof. Helena Margarida Machado da Silva Ramos

Examination Committee:

Chair person: Prof. Mario Manuel Gonçalves da Costa

Supervisor: Prof. João Eduardo de Barros Teixeira Borges

Member of the Committee: Prof. Jorge de Saldanha Gonçalves Matos

June 2016
“The wing structure of the hornet, in relation to its weight, is not suitable for flight, but he does not know this and flies anyway.”

Albert Einstein
Abstract

The purpose of this work is to study the importance of the Hydroelectric Power plant and examine the CFD modelling of a turbine with different configurations.

Therefore, it starts with a general overview of the different sources of energy mostly used in the last years, with particular attention on renewable energy sources (RES). Hydropower is the largest renewable resource used for electricity. Moving water is a powerful source of energy which is harnessed to provide clean, fast, and flexible electricity generation. Hydropower represents a reliable and domestic resource that can power millions of houses. Chapter 2 shows a brief explanation of a hydropower plant, with a small description about all the parts involved in a hydropower power plant. In this paper, I specifically examine the hydraulic turbine.

A deep explanation of turbomachines is shown in Chapter 3. Hydraulic turbines are one of the most important systems of hydropower plants. They convert almost all kinetic and potential energy of water in mechanical work, then it is transformed in electric power with a generator.

Successively the CFD analysis of a Tubular Propeller is explained. I have studied the machines using the software COMSOL Multiphysics. The analysis of two turbines is presented: 3 blades turbine and 5 blades turbine. The results obtained with the software are analysed and compared with some experimental results. After that I have studied the innovative solution of two 5 blade turbines in series.

Keywords: Micro-hydro Plant, Hydraulic turbine, CFD, Dynamic Behaviour, Tubular Propeller, Series Turbine.
# Table of contents

Abstract ................................................................................................................................................. v

List of Tables ....................................................................................................................................... iii

List of figures ......................................................................................................................................... iv

Nomenclature ......................................................................................................................................... vii

List of Acronyms .................................................................................................................................. viii

1. **Introduction** .................................................................................................................................. 1
   1.1 **Energy Overview** ................................................................................................................... 1
   1.2 **Structure of the Dissertation** ................................................................................................... 4

2. **State of the Art** ............................................................................................................................ 5
   2.1 **Hydropower Plants** .................................................................................................................... 5
       2.1.1 Types of Hydropower Plants .................................................................................................... 7
       2.1.2 Hydropower Plants’ Components ............................................................................................ 10
   2.2 **Micro-Hydro Power Plant** ......................................................................................................... 13
       2.2.1 Briefly Qualitative Analysis ................................................................................................... 13
       2.2.2 Types of micro-hydro systems ................................................................................................. 14

3. **Hydraulic turbine** .......................................................................................................................... 15
   3.1 **Basic Fundamentals** .................................................................................................................. 15
       3.1.1 Hydraulic Head Friction Losses ............................................................................................... 17
       3.1.2 Hydraulic Head Local Losses .................................................................................................. 21
   3.2 **Classifications** .......................................................................................................................... 23
   3.3 **Types of Water Turbines** ............................................................................................................ 24
   3.4 **Similarity Law** ........................................................................................................................... 27

4. **Hydro-mechanical behavior** .......................................................................................................... 29
   4.1 **Cavitation Phenomenon** ........................................................................................................... 29
   4.2 **Abrasive and Erosive wear** ...................................................................................................... 32

5. **CFD Modelling** .............................................................................................................................. 35
   5.1 **Introduction** .............................................................................................................................. 36
   5.2 **Model Description** ................................................................................................................... 36
       5.2.1 Physics ..................................................................................................................................... 36
       5.2.2 Geometry ................................................................................................................................. 38
       5.2.3 Material ................................................................................................................................... 41
       5.2.4 Boundary Conditions ............................................................................................................. 42
List of Tables

TABLE 1 - MESH DETAILS..................................................................................................................................................42
TABLE 2 - 3BT CFD RESULTS: EFFICIENCY .....................................................................................................................52
TABLE 3 - 5BT CFD RESULTS: EFFICIENCY .....................................................................................................................52
TABLE 4 - SERIES-5BT EFFICIENCY FOR Q=16 M³/H .....................................................................................................53
TABLE 5 - SERIES-5BT EFFICIENCY FOR Q=32 M³/H .....................................................................................................53
TABLE 6 – 5BT CFD RESULTS: MECHANICAL POWER FOR DIFFERENT FLOW RATE ..................................................54
TABLE 7 - AFFINITY ANALYSIS: CASE 1 .........................................................................................................................56
TABLE 8 - AFFINITY ANALYSIS: CASE 2 .........................................................................................................................56
List of figures

Figure 1 - World net electricity generation by fuel, 2010-2040 (Source: Key World Energy Statistics 2015, IEA) 1
Figure 2 - Global Temperature Anomaly (Source: GISS Surface Temperature Analysis. NASA) 2
Figure 3 - Renewable Energy Sources shares by sector in EU (RES-E: electricity; RES-H/C: heating and cooling; RES-T: transport) (Source: Eurostat, 2014b and 2015a) 3
Figure 4 - Historic contribution from RES 4
Figure 5 - Poncelet’s waterwheel with wicket gate 5
Figure 6 - Hydropower production by region (TWh) in 2015 (Source: Key World Energy Statistics 2015, IEA) 6
Figure 7 - PSH cycle 7
Figure 8 - Hydropower plant combined with wind turbines 8
Figure 9 - High potential areas for tidal resources (Source: Tidal Energy Facts, Goldman A. 2012) 9
Figure 10 - Hydropower plant’s main component 10
Figure 11 - Surge tank 11
Figure 12 - Laminar and turbulent flow 17
Figure 13 - Moody diagram 20
Figure 14 - Local head losses coefficient (Source: Hydraulic losses in pipe, Kudela H. 2010) 21
Figure 15 - Flow in a pipe: sudden enlargement 21
Figure 16 - Pipe flow: sudden contraction 22
Figure 17 - Comparison between impulse and reaction turbine configuration 24
Figure 18 - Rotors of different types of water turbine 24
Figure 19 - Pelton wheel turbine 25
Figure 20 - Vertical axes Francis turbine 26
Figure 21 - Cavitation damage on the propeller’s blades 29
Figure 22 - Mechanism of abrasive wear (Source: Tribology Engineering, G.W. Stachowiak and A.W. Batchelor, 1993) 32
Figure 23 - Mechanism of erosive wear (Source: Tribology Engineering, G. W. Stachowiak and A. W. Batchelor, 1993) 33
Figure 24 - Modify suspended concentration against turbine net head 34
Figure 25 - Experimental studies against CFD simulations 35
Figure 26 - Details of the experimental geometry (Source: Experimental characterization of a five blade tubular propeller for inline installation, Samora I. 2016) 39
Figure 27 - Whole system (Source: Experimental characterization of a five blade tubular propeller for inline installation, Samora I. 2016) 39
Figure 28 – 3BT CAD model: 3D visualization 40
Figure 29 – 5BT CAD model: 3D visualization 40
Figure 30 – Water domains 41
Figure 31 – 5BT: imported geometry (COMSOL Multiphysics) 41
Figure 32 – Efficiency trend (Source: Experimental characterization of a five blade tubular propeller for inline installation, Samora I. 2016) 54
Figure 33 – Velocity field after 2 sec: a) 3BT and b) 5BT 44
Figure 34 – Velocity field in a perpendicular section after 2 sec: 3BT and 5BT 44
Figure 35 – 3BT and 5BT CAD model, later and top view 45
Figure 36 – Shear-stress velocity after 2 sec: a) 3BT and b) 5BT 46
Figure 37 – Vorticity field: a) 3BT and b) 5BT after 2 sec 47
Figure 38 – Vorticity field in a perpendicular section after 2 sec: 3BT and 5BT 47
Figure 39 – Velocity in turbulent regime flow 48
FIGURE 86 - SERIES-SBT: VELOCITY FIELD, SLICE REPRESENTATIONS (0.0089 m³/s – 0.2 bar - 750 RPM) 94
FIGURE 87 - SERIES-SBT: VELOCITY FIELD AFTER 2 SEC (0.0089 m³/s – 0.2 bar - 1000 RPM) 95
FIGURE 88 - SERIES-SBT: VELOCITY FIELD, SLICE REPRESENTATIONS (0.0089 m³/s – 0.2 bar - 1000 RPM) 96
FIGURE 89 - SERIES-SBT: SHEAR-STRESS VELOCITY AFTER 2 SEC (0.0089 m³/s – 0.2 bar - 1000 RPM) 97
FIGURE 90 - SERIES-SBT: VORTICITY FIELD AFTER 2 SEC (0.0089 m³/s – 0.2 bar - 1000 RPM) 97
FIGURE 91 - SERIES-SBT: TURBULENT KINETIC ENERGY AFTER 2 SEC (0.0089 m³/s – 0.2 bar - 1000 RPM) 98
FIGURE 92 - SERIES-SBT: DISSIPATION RATE AFTER 2 SEC (0.0089 m³/s – 0.2 bar - 1000 RPM) 98
## Nomenclature

<table>
<thead>
<tr>
<th><strong>Greek Symbols</strong></th>
<th><strong>Roman Symbols</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>ε Absolute pipe roughness (mm)</td>
<td>A Cross section area (m²)</td>
</tr>
<tr>
<td>ε Turbulent dissipation rate (J/kg s)</td>
<td>C Hazen-Williams roughness coefficient (-)</td>
</tr>
<tr>
<td>η Overall efficiency (%)</td>
<td>D Pipe diameter (m)</td>
</tr>
<tr>
<td>ηt Turbine efficiency (%)</td>
<td>Δhₜ Total hydraulic losses (m)</td>
</tr>
<tr>
<td>ηv Volumetric efficiency (%)</td>
<td>E Energy (Wh)</td>
</tr>
<tr>
<td>ηy Hydraulic efficiency (%)</td>
<td>f Friction factor (-)</td>
</tr>
<tr>
<td>γ Unit weight of the fluid (N/m³)</td>
<td>fD Darcy-Weisbach friction factor (-)</td>
</tr>
<tr>
<td>μ Dynamic viscosity (kg/m s)</td>
<td>g Gravity acceleration (m/s²)</td>
</tr>
<tr>
<td>μt Turbulent viscosity (kg/m s)</td>
<td>hf Friction head loss (m)</td>
</tr>
<tr>
<td>ν Kinematic Viscosity (m²/s)</td>
<td>h Local head loss (m)</td>
</tr>
<tr>
<td>π Pi (⁻)</td>
<td>Hn Net head (m)</td>
</tr>
<tr>
<td>ρ Water density (kg/m³)</td>
<td>Hp Pump head (m)</td>
</tr>
<tr>
<td>σth Thoma parameter (⁻)</td>
<td>H Total head (m)</td>
</tr>
<tr>
<td>τ Shear stress (Pa)</td>
<td>K Local loss coefficient (⁻)</td>
</tr>
<tr>
<td>ω Rotation speed (rpm)</td>
<td>L Pipe length (m)</td>
</tr>
<tr>
<td></td>
<td>Ms Mach number (⁻)</td>
</tr>
<tr>
<td></td>
<td>N Rotation speed (rpm)</td>
</tr>
<tr>
<td></td>
<td>nt Specific speed (⁻)</td>
</tr>
</tbody>
</table>
List of Acronyms

3BT Three Blade Turbine
5BT Five Blade Turbine
BDF Backward differentiation formula
CFD Computation Fluid Dynamics
DOE Department of Energy
EEA European Environmental Agency
EPRI Electric Power Research Institute
GHG Greenhouse Gas
GO Gate Openings
HPW Hydrostatic Pressure Wheel
IDC Inner dead centre
IEA International Energy Agency
NREAPs National Renewable Energy Action Plants
NPSH Net Positive Suction Head
ODC Outer dead centre
ODE Ordinary Differential Equation
PHS Pumped Hydro Storage
RANS Reynolds averaged Navier-Stokes equations
RES Renewable Energy Sources
TKE Turbulent kinetic energy
1. Introduction

1.1 Energy Overview

The majority of the electricity generated in the world today comes from fossil fuels. Accounting for approximately 67% of the world’s net electricity estimated in 2010 is generated from fossil fuel, while nuclear and renewable sources account for roughly 12% and 21%, respectively (Figure 1). By 2040, world population is expected to grow from 7 to 8.8 billion people, mainly from developing economies, such as India and China, while the world’s real gross domestic product rises by an average of 4% per year from 2010 to 2040.

![Figure 1 - World net electricity generation by fuel, 2010-2040 (Source: Key World Energy Statistics 2015, IEA)](image)

These two factors are the key drivers behind the increase of electricity consumption, being in turn offset by efficiency gains from new appliance standards and investments in energy efficient equipment. With current policies, fossil fuels are estimated to still have the biggest share of the world’s net electricity generation by 2040, with roughly 62%, with renewable and nuclear sources increasing to 25% and 13%, respectively. 1

The present energy economy based on fossil fuels is at serious risk due to a series of factors:

- The continuous increase in demand for oil;
- The depletion of non-renewable resources;
- The increase of the content of carbon dioxide gas in the atmosphere, causing greenhouse effect;

---

1 Key World Energy Statistics 2015, IEA
The global warming phenomena, as the following picture shows (Figure 2).

![Reconstructed Temperature](image)

**Figure 2 - Global Temperature Anomaly (Source: GISS Surface Temperature Analysis. NASA)**

Based on the analysis of the information reported above, the European Union defined targets for GHG emissions to be met by 2020, renewable energy and energy efficiency. “Europe 2020” is a 10-year strategy proposed by the European Commission on 3rd March 2010 for advancement of the economy of the European Union.

The 2020 strategy is a set of binding legislation to ensure the EU meets its climate and energy targets for the year 2020. The package sets three key targets:

- **GHG emissions.** These were already 19.8% below 1990 levels in 2013, very close to the 20% reduction target set for 2020. The EU may achieve a GHG reduction of 24% below 1990 levels by 2020 with the current measures in place, according to the latest projections from Member States. Additional measures (currently planned by Member States) could further reduce emissions to 25% below 1990 levels. Most of the savings in GHG emissions are expected to take place under the EU ETS, which today represents about 45% of total EU emissions. Approximated estimates of 2014 GHG emissions reported by Member States indicate that GHG emissions decreased significantly in 2014. Compared to 1990 levels, the reduction reached 23% (24% if international aviation is excluded). The year 2014 was exceptionally warm in almost all parts of Europe, resulting in a markedly low heating energy demand compared to 2013. This 2014 level is significantly lower than that anticipated by Member States in their projections.

- **Renewable energy.** The steady deployment of RES in the EU's energy mix continues. The consumption of renewable energy continued to increase in 2013, standing at 15% of gross final energy consumption and getting closer to the 20% target for 2020. The 2013 share is higher than indicative levels set for that year in both the RED and Member States' national renewable energy action plans (NREAPs). The 2020 target could be attained if Member States can sustain the speed at which they have been developing RES, so far. However, as we approach 2020, the trajectories...

---

2 Trends and projections in Europe 2015, Tracking progress towards Europe’s climate and energy targets, EEA
for meeting the national targets become steeper, and more costly projects will have to be developed, while market barriers persist in several Member States. According to approximated estimates from the EEA, the EU RES share further increased in 2014 and remained above the indicative share set in the RED for the two-year period from 2013 to 2014.

- **Energy efficiency.** The EU is reducing its energy consumption. Since 2005, the EU's primary and final energy consumption has been decreasing at a pace which, if sustained until 2020, would be sufficient for the EU to meet its 20% energy efficiency target. Again, the pace might be difficult to sustain because European legislation implementation remains weak in several Member States. Considering EEA preliminary estimates, primary energy consumption further decreased in 2014 in most Member States. This decrease can be partly explained by the warmer temperatures in 2014, compared to 2013.

The potential of renewable energy sources (RES) is enormous as they can meet in principle meet many times the world’s energy demand. RES such as biomass, wind, solar, hydropower, and geothermal can provide sustainable energy services, based on the use of routinely available, indigenous resources. A transition to renewables-based energy systems is looking increasingly likely as the costs of solar and wind power systems have dropped substantially in the past 30 years, and continue to decline, while the price of oil and gas continue to fluctuate.

Renewable energy sources currently supply somewhere between 15% and 20% of world’s total energy demand. The supply is dominated by traditional biomass, mostly fuel wood used for cooking and heating, especially in developing countries in Africa, Asia and Latin America. A major contribution is also obtained from the use of large hydropower with nearly 20% of the global electricity supply being provided by this source. New renewable energy sources (solar energy, wind energy, modern bio-energy, geothermal energy, and small hydropower) are currently contributing about 2%. Figure 4 shows the contribution of each type of RES in the last few years.
It is becoming clear that future growth in the energy sector is primarily in the new regime of renewable, and to some extent natural gas-based systems, and not in conventional oil and coal sources. Financial markets are awakening to the future growth potential of renewable and other new energy technologies, and this is like a new approach for the economic reality of truly competitive renewable energy systems.

### 1.2 Structure of the Dissertation

The present dissertation is divided into five chapters. In short, the first chapters, from two to five, correspond to the theoretical part. In this part a briefly description of all the structures present in each hydropower plant will be given; also in this first part (chapter 3) an analysis and description of micro-hydropower stations is made, whereas chapter 4 describes the physics of hydraulic turbines. Chapter 5 presents a discussion of the dynamic behaviour for hydraulic machines; it shows the physics of cavitation and the erosive and abrasive wear.

The second part of this dissertation is focused on CFD modelling. In chapter 5 the basic steps to model a tubular propeller inside a pressurized pipe are shown; moreover, chapter 6 presents the analysis of the results obtained from the software COMSOL Multiphysics.

The models analysed are two: one of a 5 blades tubular turbine and one of a 3 blades tubular turbine. Then, an analysis of the results obtained will be made.
2. State of the Art

2.1 Hydropower Plants

Hydropower was already used in China at least 2000 years ago; the waterwheel was invented in ancient Greece and Rome; in the year 13 B.C., the Roman engineer and writer Marcus Vitruvius Pollio described a grain mill driven by a waterwheel and a cogwheel gear. The variety of waterwheel applications increased greatly through the Middle Ages. Around 1500, the waterwheel was the most important tool for power generation in Europe and elsewhere. In the 16th century, Leonardo da Vinci made some sketches that are almost recognizable as water turbines as we know them today. After 1770, waterwheels were consistently improved, and wheels made from cast iron or even sheet metal began to appear.

A wicket gate proposed by Euler in 1754 forced the flow in a certain direction, thus reducing the hydraulic losses when entering the runner. As a result, the first real turbines appeared.

Finally, by 1826, a detailed theory of waterwheels existed and some types of speed control were proposed in publications. The first step toward a turbine was taken in France by Jean Victoire Poncelet (1788-1867). In 1825, he built a waterwheel with curved blades. The curved blades effectively reduced internal hydraulic losses; in addition, Poncelet invented an installation to change the flow (and thus both speed and torque) by then resembled a wicket gate. As shown in the Figure 5, it was adjusted by hand.

The name turbine was probably first used in 1824 by the Frenchman M. Burdin. In the United States, James B. Francis (1815-1892) improved upon some inventions of turbines. This resulted in a turbine with spiral casing, a circular wicket gate (not yet adjustable) placed around the periphery of the runner, and a draft tube. About 1860, the blades of such wicket gates were made adjustable simultaneously, and gradually the many other types of turbines became speed controlled by centrifugal regulators.

However, only in the late 19th century, hydropower became a source for generating electricity. In the 1800s the water wheel was often used to power machines such as timber-cutting saws in European and American factories. More importantly, people realized that the force of water falling from a height would turn a turbine connected to a generator to produce electricity. In 1880, a dynamo driven by a water turbine was used to provide light to a theatre in Grand Rapids, Michigan, and in 1881, a dynamo connected to a turbine in a flour mill provided street lighting at Niagara Falls, New York; both of which used direct current (DC) technology. The breakthrough of alternating current, allowed power to be transmitted at longer distances and ushered in the first U.S. commercial installation of an alternating current (AC) hydropower plant at the Redlands
Power Plant in California in 1893. The Redlands Power Plant utilized Pelton waterwheels driven by water taken from the nearby Mill Creek and a 3-phase generator which ensured consistent power delivery.³

Aside from a plant for electricity production, a hydropower facility consists of a water reservoir enclosed by a dam whose gates can open or close depending on how much water is needed to produce a particular amount of electricity. Man-made waterfalls dams were constructed throughout the 1900s in order to maximize this source of energy. It is clear from the following figure that the Hydropower generation had a considerable increase during the last 10 years.

![Hydropower production by region (TWh) in 2015](image)

*Figure 6 - Hydropower production by region (TWh) in 2015 (Source: Key World Energy Statistics 2015, IEA)*

Hydro resources are also widely distributed compared to fossil and nuclear fuels and can help provide energy independence for countries without fossil fuel resources. There is also widespread activity in developing small, mini and micro hydro plants. At least forty countries, particularly in Asia and Europe, have plants under construction and even more have plants planned. China, Brazil, Canada, Turkey, Italy, Japan and Spain all have plans for more than 100 MW of new capacity.

2015 Key Water Power Program and National Laboratory is committed from U.S. Department of Energy (DOE) to developing and deploying a portfolio of innovative and efficient technologies solution to generate clean power from water resources across the United States. The Water Power Program’s research efforts focus on improving the performance, lowering the cost, and accelerating the deployment of hydropower and marine and hydrokinetic (MHK) technologies. The project is focused on developing the next generation of water power tools and technologies, while jump-starting the private sector innovation critical to the country’s long-term economic growth, energy security, and international competitiveness.⁴

---

³ EERE, US Department of Energy.
⁴ 2015 Key Water Power Program and National Laboratory Accomplishments, U.S DOE.
2.1.1 Types of Hydropower Plants

There are four types of hydropower facilities: Conventional (Dams), Pumped Storage, Run-of-the-River and Tide.

Conventional Plant

The most common type of hydroelectric power plant is an impoundment facility. An impoundment facility, typically a large hydropower system, uses a dam to store river water in a reservoir. Water released from the reservoir flows through one or more turbines, spinning it, which in turn activates a generator to produce electricity. The power extracted from the water depends on the volume and on the difference in height between the source and the water's outflow. This height difference is called the head. A large pipe (the "penstock") delivers water from the reservoir to the turbine. The water may be released either to meet changing electricity needs or to maintain a constant reservoir level.

Pumped Storage

The most widely used form of bulk-energy storage is currently pumped-storage hydropower (PSH), which uses the simple combination of water and gravity to capture off-peak power and release it at times of high demand. Pumped-hydro facilities typically take advantage of natural topography, and are built around two reservoirs at different heights. As the figure shows, off-peak electricity, usually during the night, is used to pump water from the lower to the higher reservoir, turning electrical energy into gravitational potential energy. When power is needed, water is released back down to the lower reservoir, spinning a turbine and generating electricity.

PSH counts for more than 99% of bulk storage capacity worldwide: around 127.000 MW, according to the Electric Power Research Institute (EPRI), the research arm of America’s power utilities. Hydroelectric plants are more efficient at providing for peak power demands during short periods than are fossil-fuel and nuclear power plants are. Pumped storage is a method of keeping water in reserve for peak period power demands by pumping water that has already flowed through the turbines back up a storage pool above the power plant at a time when customer demand for energy is low, such as during the middle of the night. The water is then allowed to flow back through the turbine-generators at times when demand is high and a heavy load is placed on the system.
The reservoir acts much like a battery, storing power in the form of water when demands are low and producing maximum power during daily and seasonal peak periods. An advantage of pumped storage is that hydroelectric generating units are able to start up quickly and make rapid adjustments in output. They operate efficiently when used for one hour or several hours. Moreover, because pumped storage reservoirs are relatively small, construction costs are generally low compared with conventional hydropower facilities.⁵

In 2009, world pumped storage generating capacity was 104 GW, while other sources claim 127 GW, which comprises the vast majority of all types of utility grade of hydropower electric storage. The EU had 38.3 GW net capacity (36.8% of world capacity) out of a total of 140 GW and representing 5% of total net electrical capacity in the EU. Japan had 25.5 GW net capacity (24.5% of world capacity). In 2010 the United States had 21.5 GW of pumped storage generating capacity (20.6% of world capacity). PHS generated (net) 5.501 GWh of energy in 2010 in the US because more energy is consumed in pumping than is generated.

PHS could be a really interesting solution to store other renewable energy as wind energy. The scope of this solution is to combine multiple source to deliver non-intermittent electric power. Therefore, when wind energy is combined with a pump-hydro system, several advantages can be achieved: the plant will be completely environmental friendly and these storage systems can be used to regulate the energy delivery and avoid the problem of the wind high variability. Moreover, when there is a variable tariff, it is possible to achieve significant economical benefits by using optimal pumping/turbine schedules.

**Figure 8 - Hydropower plant combined with wind turbines**

---

**Run-of-the-River**

Run-of-the-river hydroelectric stations are those with small or no reservoir capacity, so that only the water coming from upstream is available for generation at that moment, and any oversupply must pass unused. A constant supply of water from a lake or existing reservoir upstream is a significant advantage in choosing sites for run-of-the-river.

---

In the United States, a run of the river hydropower could potentially provide 60,000 MW, about 13.7% of total use in 2011 if continuously available. The prospects for the future development of run-of-the-river projects seem relatively bright. As the renewable energy sector continues to grow, this form of hydropower looks set to emerge as an increasingly important resource.

**Tide**

Historically, tide mills have been used both in Europe and on the Atlantic coast of North America. The incoming water was contained in large storage ponds, and as the tide went out, it turned waterwheels that used the mechanical power it produced to mill grain. The earliest occurrences date from the Middle Ages, or even from Roman times.

![Figure 9 - High Potential Areas for Tidal Resources (Source: Tidal Energy Facts, Goldman A. 2012)](image)

A tidal power station makes use of the daily rise and fall of ocean water due to tides; Tidal power is a form of hydropower that converts the energy obtained from tides into useful forms of power, mainly electricity.

Although not yet widely used, tidal power has potential for future electricity generation. Tides are more predictable than wind energy and solar power.

Among sources of renewable energy, tidal power has traditionally suffered from relatively high costs. The traditional tidal electricity generation involves the construction of a barrage across an estuary to block the incoming and outgoing tide. Tidal range may vary over a wide range (4.5-12.4 m) from site to site. However, a tidal range of at least 7 m is required for economical operation and for sufficient head of water for the turbines.

A 240 MW facility has been operating in La Rance river estuary on the northern coast of France since 1966, The La Rance generating station has been in operation since 1966 and has been a very reliable source of electricity for France. The La Rance
tidal power facility, built between 1961 and 1966, involved the construction of a 145.1m long barrage with six fixed wheel gates and a 163.6m-long dyke. The basin area covered by the plant is 22 km$^2$. Power is produced through 24 reversible bulb turbines with a rated capacity of 10 MW each.\textsuperscript{6}

Elsewhere there is a 20 MW experimental facility at Annapolis Royal in Nova Scotia (Canada), and a 0.4 MW tidal power plant near Murmansk in Russia. UK has several proposals underway. Moreover, there are a number of stations in China since 1977, totalling 5 MW.\textsuperscript{7}

However, many recent technological developments and improvements, both in design (e.g. dynamic tidal power, tidal lagoons) and turbine technology (e.g. new axial turbines), indicate that the total availability of tidal power may be much higher than previously assumed, and that economic and environmental costs may be brought down to competitive levels.

\subsection*{2.1.2 Hydropower Plants' Components}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{images/hydropower_plant.png}
\caption{Hydropower Plant's Main Component}
\end{figure}

The water behind the dam flows through an intake and pushes against blades of the turbine, causing them to turn. The turbine spins a generator to produce electricity. The amount of electricity that can be generated depends on how far the water drops and how much water moves through the system.

\textbf{Forebay Tank (Reservoir)}

As the name suggests forebay is an enlarged body of water in front of intake. The reservoir acts as forebay when penstock takes water directly from it. The forebay tank forms the connection between the channel and the penstock. The main purpose is to allow the last particles to settle down before the water enters the penstock. Depending on its size it can also serve as a

\begin{itemize}
\item \textsuperscript{6} “Tidal giants - the world’s five biggest tidal power plants”. 11 April 2014
\item \textsuperscript{7} 2015 Ocean Energy Council Inc., http://www.oceanenergycouncil.com/ocean-energy/tidal-energy/
\end{itemize}
reservoir to store water. The forebay temporarily stores water for supplying the same to the turbines. In front of the gates trash racks are provided to prevent debris, trees, etc., from entering into the penstock. Rakes are also provided to clean the trash racks at intervals.

**Intake Conduit and Control Gates**

The gates are built on the inside of the dam. The water from reservoir is released and controlled through these gates. These are called inlet gates because water enters the power generation unit through these gates. When the control gates are opened the water flows due to gravity through the penstock and towards the turbines. The water flowing through the gates possesses potential as well as kinetic energy.

The intake of a hydro scheme is designed to divert a certain part of the river flow. This part can go up to 100 % as the total flow of the river is diverted via the hydro installation. The following points are required for an intake: The desired flow must be diverted; The peak flow of the river must be able to pass the intake and weir without causing damage to them; As less as possible maintenance and repairs; It must prevent large quantities of loose material from entering the channel; It must have the possibility to remove piled up sediment.

**The Penstock**

The penstock is the long pipe that carries the water flowing from the reservoir towards the power generation unit. The water in the penstock possesses kinetic energy due to its motion and potential energy due to its height. The total amount of power generated in the hydroelectric power plant depends on the height of the water reservoir and the amount of water flowing through the penstock. The amount of water flowing through the penstock is controlled by the control gates.

The penstock often constitutes a major expense in the total micro hydro budget, as much as 40 % is not uncommon in high head installations, and it is therefore worthwhile optimising the design. Head loss due to friction in the pipe decrease dramatically with increasing pipe diameter. Conversely, pipe costs increase steeply with diameter. Therefore, a compromise between cost and performance is required.

**Surge Tank**

A surge tank is a storage reservoir fitted at some opening made on the penstock to receive the rejected flow when the pipeline is suddenly closed by a valve fitted at its steep end. A surge tank, therefore, relieves the pipe line of excessive pressure produced due to its closing, thus eliminating the positive water hammer effect. When the turbine gates are closed, the moving water has to turn around; a surge tank would then act as a receptacle to store the rejected water and thus avoids water

![Figure 11 - Surge Tank](image-url)
hammer. On the other hand, when there is an immediate energy demand, the turbine can produce more power re-opening the gates in proportion to the increased load, thus, making it necessary to supply more water.

**Turbines and Generators**

Turbines convert hydraulic energy into mechanical energy. The mechanical energy developed by a turbine is used to run an electric generator. The hydraulic turbine directly coupled to the shaft of the generator. The generator develops electric power.

When water falls on the blades of the turbine the kinetic and potential energy of water is converted into the rotational motion of the blades of the turbine. The rotating blades causes the shaft of the turbine to also rotate. The turbine shaft is enclosed inside the generator.

There are various types of water turbines such as Kaplan turbine, Francis turbine, Pelton wheels etc. The type of turbine used in the hydroelectric power plant depends on the height of the reservoir, on the quantity of water and on the total power generation capacity.

**Powerhouse**

The purpose of the powerhouse is to support and protect the hydraulic and electrical equipment. The powerhouse usually is usually divided into two parts as follows:

- The substructure to support the equipment and to provide the necessary water-ways. The substructure may form an integral part of the dam and intake structure. In other cases, the substructure may be remote from the dam, the dam intake and power house being entirely separate structures. The substructure is built exclusively of concrete and is enforced with steel where necessary.

- The superstructure, which that is the generating room. It is the main portion of the powerhouse, contains the main units and their accessories, and usually there is a power or hand operated overhead crane which spans the width of the power house. The switch board and operating stand are usually near the middle of the station, either at floor level or, for better visibility, on the second floor or at a level above the main floor. Usually an auxiliary bay or section of the power house will be required upstream from the main units for the switches, bus connections, and outgoing lines. If transformers are located inside the station, these will also be in the auxiliary bay, commonly at floor level and shut off the main floor by steel doors or shutters. A travelling crane is an important part of the power house equipment. In fixing the elevation of the crane rail above the floor, it is essential that sufficient headroom be provided for lifting and carrying along any of the various machine parts.
2.2 Micro-Hydro Power Plant

2.2.1 Briefly Qualitative Analysis

Globally, hydropower plants are the largest source clean electricity. The 1.000 GW of installed hydro capacity provided about 16% of the world’s electricity in 2013; most of this is from large-scale systems. However, in the last years, a lot of researches are developing to improve the efficiency of small-hydro systems. In 2009, the global capacity of small-scale hydro was estimated at about 60 GW. The main micro-hydro programmes are in mountainous countries, such as Nepal (around 2.000 schemes, including both mechanical and electrical power generation) and other countries in the Himalayas. There are also many schemes in South America, particularly in countries along the Andes such as Peru and Bolivia. China has seen the main growth in use of small-scale hydro in recent years.8

Small-scale hydropower is both an efficient and reliable form of energy. Small hydropower can be classified in mini, micro or pico, depending on the output power and on the type of the adopted scheme. There are not yet globally accepted boundaries to define these classes, but micro-hydro typically refers to schemes below 100 kW, while pico-hydro usually produces less than 5 kW.

Micro hydro is a type of hydroelectric power that typically produces electricity using the natural flow of water (Run-of-the-River hydroelectric power plant). Because of the amount of energy produced is quite low, most of the micro-hydropower systems are used by homeowners and small business owners, including farmers and ranchers.

Communities choose micro hydro systems for several reasons. Environmental motivations are very common, in fact, by reducing the need to cut down trees for firewood and increasing farming efficiency; micro-hydro power has a positive effect on the local environment. Micro hydro systems can generate a large amount of energy out of a small water flow with minimal impact. These systems can be low-wattage while generating enough energy to make a big dent in a typical home’s energy use, and, in off-grid systems, even minimize or eliminate the need for having batteries. This is the main use of this type of system: generating energy from low-head water course to supply off-grid houses.

Moreover, micro-hydro power can also be used to recharge batteries. People can use these convenient sources of electricity to fuel anything, from workshop machines to domestic lighting, and there are no expensive connection costs. The batteries can be charged at a station in the village, thus providing to the local community clean and renewable energy.

These systems can also make the users entirely independent of the grid, but anyway they can be connected to the grid, allowing you to “sell back” surplus electricity for a credit and providing backup when the utility fails, giving you the best of

---

8 http://www.ashden.org/micro-hydro
both worlds. Additional benefits include that maintenance is done at ground level and the system’s production is around the clock.

There are several companies that provide to install this type of system where electric energy is need. *Practical Action*\(^9\) is one of them; they have developed micro-hydro systems with communities in Peru, Zimbabwe, Sri Lanka, and Kenya. These systems, which are designed to operate for a minimum of 20 years, are usually ‘run-of-the-river’ systems. *ONE* is a campaigning and advocacy organization of more than seven million people around the world taking action to end extreme poverty and preventable disease, particularly in Africa. This organization started the project *Micro hydro power lights up life in Africa*\(^10\) to find possible solutions to provide electric energy for the world’s poorest people as part of *The Energy Poverty Challenge*\(^11\).

### 2.2.2 Types of micro-hydro systems

As with wind and solar systems, hydro-electric systems can be divided into four configurations:

- **On-grid without batteries**: This is a simple and efficient system that sends any surplus energy back into the grid to be credited to you for use at other times. These systems typically do not provide backup for utility outages.

- **On-grid with batteries**: This system type also sells back surplus electricity, but also provides backup during utility outages. The amount of backup will be determined by the system’s capacity and the battery size.

- **Off-grid without batteries**: This configuration is generally for larger, AC-generating systems. The peak load capacity (how many things you can operate at once) is determined by the hydro system’s peak generating capacity. This configuration is generally not used for systems that generate less than about 2 kW.

- **Off-grid with batteries**: This is the most common off-grid option, and is similar to off-grid solar- or wind-electric systems. The charging source puts energy into a battery bank, while loads are run from the batteries—directly, if DC; via an inverter, if AC.

Micro hydro systems can power most, if not all, electrical loads, depending on the size of the resource. The smallest systems may only provide for lighting, electronics, and basic refrigeration. But with sufficient head and flow, these systems can run heating and cooling systems, tools, and even commercial equipment in a modern, on-grid home, ranch, or business.

---

9 [http://practicalaction.org/micro-hydro-power](http://practicalaction.org/micro-hydro-power)
11 [http://www.one.org/us/2012/05/14/the-energy-poverty-challenge/](http://www.one.org/us/2012/05/14/the-energy-poverty-challenge/)
3. Hydraulic turbine

3.1 Basic Fundamentals

The total energy at a given point in a fluid is the energy associated with the movement of the fluid, plus energy from pressure in the fluid, plus energy from the height of the fluid relative to an arbitrary datum. To evaluate the power that the water turbine produces in a hydropower plant, it is important to evaluate the work when the water goes through its.

First of all, considering the first thermodynamic principle between the reservoir (1) and the outlet section of the turbomachinery (2):

\[
W = \frac{p_1 - p_2}{\rho} + \frac{V_1^2 - V_2^2}{2g} + (z_1 - z_2) - W_f
\]

where \( W \) is the work per unit mass, \( p \) is the pressure of the water, \( \rho \) is the density of the water, \( V \) is the velocity of the water, \( z \) is the height in the vertical direction, \( g \) is the gravity acceleration, \( W_f \) are the friction losses.

Head is expressed in units of height such as meters. In fluid dynamics, head is a concept that relates the energy in the fluid to the height of an equivalent static column of that fluid; if the fluid is water 1 bar corresponds to 10,197 mWc. Head is equal to the fluid's energy per unit weight.

From Bernoulli's Principle for a steady state the sum of all forms of energy along a streamline is the same at all points on that streamline. This requires that the sum of kinetic energy, potential energy and internal energy remains constant. Thus an increase in the speed of the fluid (implying an increase in both its dynamic pressure and kinetic energy) occurs with a simultaneous decrease in its static pressure, potential energy and internal energy. From this principle we are able to compute the total head of a turbine.

There are four components in the formulation of the total head of a turbine. The subscript 1 indicate the inlet section of the turbine, the subscript 2 indicate the outlet section of the turbine:

- \( \frac{p_1 - p_2}{\rho g} \): Pressure head is due to the static pressure, the internal molecular motion of a fluid that exerts a force on its container;

- \( \frac{V_1^2 - V_2^2}{2g} \): Velocity head is due to the bulk motion of a fluid (kinetic energy);

- \( (z_1 - z_2) \): Elevation head is due to the fluid’s weight, the gravitational force acting on a column of fluid;

- \( W_f \): Resistance head (or friction head or Head Loss) is due to the frictional forces acting against a fluid's motion.
Thus the total head at the inlet section (1) of the turbine is:

\[ H_1 = \frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 \]  

(3.2)

Meanwhile at the outlet section (2):

\[ H_2 = \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \]  

(3.3)

The total head or hydraulic head is defined as:

\[ H_t = H_1 - H_2 \]  

(3.4)

Thus, the total work per unit mass produced by the turbine is:

\[ W = g[H_1 - H_2] \]  

(3.5)

\[ W = g H_t \]  

(3.6)

From this relation it is clear the Bernoulli’s principle: if there is not work supplied and there are no losses, the total head is constant. However, of course not all the hydraulic head can be convert into energy, there are always losses.

It is possible now to define the Net Head:

\[ H_n = H_t - \Delta h_{ls} \]  

(3.7)

The scalar quantity $\Delta h_{ls}$ is called hydraulic head losses. The concentrated (local) and friction (distributed) losses of head cause difference in the Bernoulli’s theorem calculated in two different sections. So when the viscosity of the fluid is taken into account total energy head $H_t$ is no longer constant along the pipe. In direction of flow, due to friction cause by viscosity of the fluid we have:

\[ \frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 > \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \]  

(3.8)

Then to restore the equality we must add some scalar quantity to the right side of this inequality:

\[ \frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + z_2 + \Delta h_{ls} \]  

(3.9)

or

\[ H_n = H_t - \Delta h_{ls} \]  

(3.10)
The hydraulic loss between two different cross section along the pipe is equal to the sum of friction head losses and local head losses:

$$\Delta h_{ls} = h_f + h_l$$

(3.11)

The $h_f$ is the effective head used to create power. The power produced by a turbine depends entirely on the flow rate of the water, vertical head that the water falls and the acceleration of water due to gravity through following equation:

$$P_t = \eta_t \gamma Q H_n$$

(3.12)

### 3.1.1 Hydraulic Head Friction Losses

The friction losses or distributed losses mainly depend on the flow velocity. Friction loss, which is due to the shear stress between the pipe surface and the fluid flowing within, depends on the conditions of flow and the physical properties of the system. These conditions can be encapsulated into a dimensionless number, Reynolds number:

$$Re = \frac{\rho V D}{\mu}$$

(3.13)

or

$$Re = \frac{V D}{\mu}$$

(3.14)

Where $\rho$ is the water density (kg/m$^3$), $V$ is the mean velocity of the fluid (m/s), $D$ is the diameter of the pipe (m), $\mu$ is the dynamic viscosity of the fluid (kg/m s), $\nu$ is the kinematic viscosity of the fluid ($\nu=\mu/\rho$) (m$^2$/s).

In closed conduit, **laminar flow** occurs when the Reynolds number is below a critical value of approximately 2040, through the transition range is typically between 2000 and 4000. Shear flows undergo a sudden transition from laminar to turbulent motion as the velocity increases, and the onset of turbulence radically changes transport efficiency and mixing properties.

![Figure 12 - Laminar and Turbulent Flow](image)

When laminar flow occurs, the friction factor can be computed using the following equation:
**Hagen–Poiseuille** equation:

\[ h_f = \frac{32 \mu L V}{\gamma D^2} \quad (3.15) \]

or

\[ h_f = \frac{8 \mu L Q}{\pi \rho g R^4} \quad (3.16) \]

or

\[ h_f = \frac{128 \mu L Q}{\pi \rho g D^4} \quad (3.17) \]

Where \( L \) is the length of the pipe (m), \( R \) is the radius of the pipe (m).

The assumptions of the equation are that the fluid is incompressible and Newtonian; the flow is laminar through a pipe of constant circular cross-section that is substantially longer than its diameter and there is no acceleration of fluid in the pipe.

**Darcy-Weisbach relation**:

\[ h_f = f_D \frac{L V^2}{D 2g} \quad (3.18) \]

From the relation between the wall shear stress and the head loss and the energy equation between two section:

\[ h_f = \frac{4 \tau_w L}{\gamma D} \quad (3.19) \]

where \( \tau_w \) is the shear stress:

\[ \tau_w = \frac{4 \mu V}{r} \quad (3.20) \]

Thus from these equations, the friction factor for laminar flow is:

\[ f_D = \frac{64}{Re} \quad (3.21) \]

When Reynolds number is over 4000 turbulent flow occurs. When flow becomes turbulent the fluid undergoes irregular fluctuations, or mixing, in contrast to laminar flow, in which the fluid moves in smooth paths or layers. In turbulent flow the speed of the fluid at a point is continuously undergoing changes in both magnitude and direction.
To compute the losses for turbulent regime we may use one of the following relations:

- **Blasius equation:**

  \[ f = \frac{0.316}{Re^{0.25}} \]  
  (3.22)

  This relation may be used for smooth pipes and only for Reynolds’ Number up to \(10^5\).

- **Moody/Nikuradse diagram:**

The first engineer to study the diagram was Lewis Moody, in 1944. His diagram attempts to explain how big a pump you have to put on one end of a pipe to make water flow given roughness in the pipe. This results in the first curiosity in the Moody diagram - the Reynolds number uses pipe diameter for the length scale, and not flow depth. This also means the relative roughness uses pipe diameter. The Moody diagram works well, except for one issue — it was originally designed for pipes, and deals principally with rust and corrosion for the roughness elements.

Johann Nikuradse tried to find out more too, before Moody. Even Nikuradse (1933) was gluing sand to the inside of glass pipes and measuring the frictional velocity loss. Nikuradse, it wasn’t until 1958 that Rouse brought his data to light. Curiously, even though a Nikuradse diagram appears in Rouse’s 1958 book, most people ascribe the modern Nikuradse diagram to Brownlie (1983). Remember that Nikuradse’s diagram has the same issues Moody’s does—you have to use \(D=4R_h\). Next issue—Nikuradse shows data for uniform grain size; when the grain size is mixed, the larger roughness elements dominate, and you end up with a different curve. Nikuradse conducted experiments with water in circular pipes to study the laminar and turbulent regions, covering Reynolds from 600 to \(10^6\). By using pipes with different dimensions and different roughness, he produced a set of experimental results of \(f\) and \(Re\) for a range of relative roughness of \(1/30\) to \(1/1014\). He defined the relative roughness as the ratio between the sand grain size and the pipe diameter (\(\varepsilon/D\)).

He discovered that there are three ranges:

- In range I, for small Reynolds number the resistance factor is the same for rough as for smooth pipes. The roughening line in entirely within the laminar layer for this range.

- In range II (transition range), an increase in the resistance factor was observed for an increasing Reynolds number. The thickness of the laminar layer is here of the same order of magnitude as that of the projections.

- In range III, the resistance factor is independent of the Reynolds number (quadratic law of resistance).
The Hazen-Williams formula is an empirical equation. This equation uses the coefficient C to specify the pipes roughness, which is not based on a function of the Reynolds number. Thus, this equation has the advantage that the coefficient C is not a function of the Reynolds number, but it has the disadvantage that it is only valid for water. Moreover, it does not account for the temperature or viscosity of the water.

\[
h_f = \frac{6.78 L}{D^{1.65}} \left(\frac{V}{C}\right)^{1.85}
\]  

(3.25)
The coefficient C varies from about 70 up to 150. It depends on pipe diameter, material and age.

The Hazen William formula has now become adopted through the world as the pressure loss formula to use for the hydraulic design of fire sprinkler systems and in almost all cases the use of the Hazen-William formula will provide adequate answers.

### 3.1.2 Hydraulic Head Local Losses

The local losses or minor losses may raise at pipe entrance or exit, sudden or gradual expansion or contraction, bends, elbows, and valves. The most common relation used to compute the local head losses is:

\[
h_l = K_L \frac{V^2}{2g}
\]  

(3.26)

Where \(K_L\) is the local loss coefficient. It is tabulated for the most common pipe components.

![Figure 14 - Local head losses coefficient (Source: Hydraulic losses in pipe, Kudela H. 2010)](image)

The enlargement or contraction loss is computed from the Borda–Carnot equation. The Borda–Carnot equation is applied to the flow that goes through a sudden contraction or expansion in a horizontal pipe.

![Figure 15 - Flow in a Pipe: sudden enlargement](image)
It is clear that velocity decrease from position 1 to 2 and therefore the pressure increase. At position 3 turbulent eddies are formed, which gives rise to a local energy loss.

Applying the momentum equation between the section 1 and 2:

\[ p_1 A_1 - p_2 A_2 = \rho Q (V_2 - V_1) \]  

(3.27)

The continuity equation \((Q = A_2 V_2)\) is used to express the volume flow rate \(Q\) as product of cross section area \(A\) and mean velocity \(V\). The local head losses \(h_l\) may be founded applying the energy balance between the sections 1 and 2:

\[ \frac{p_1}{\rho g} + \frac{V_1^2}{2g} = \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + h_l \]  

(3.28)

or, alternatively,

\[ h_l = \frac{V_1^2 - V_2^2}{2g} - \frac{p_2 - p_1}{\rho g} \]  

(3.29)

Rearranging the momentum equation and the energy balance relation, the local head losses are:

\[ h_l = \frac{(V_1 - V_2)^2}{2g} \]  

(3.30)

The continuity equation may be now use again in order to express the results obtained only in terms of the two areas and the inlet mean velocity. Since the volume flow rate between the two section is constant, hence \(V_2 = V_1 A_1 / A_2\).

Finally, the local head losses is:

\[ h_f = \frac{V_1^2}{2g} \left(1 - \frac{A_1}{A_2}\right)^2 \]  

(3.31)

Instead, when along the pipe there is a sudden contraction, experiment indicate that the contraction of the flow area is generally about 40%.

If the energy losses between section 1 and 3 are negligible, then the remaining head losses occurs between 3 and 2. Substituting \(A_1 = 0.6 A_2\).
\begin{equation}
    h_f = \frac{(V_2/0.6)^2}{2g} \left(1 - \frac{0.6 A_2}{A_2}\right)^2
\end{equation}

or

\begin{equation}
    h_f = \frac{0.44 V_2^2}{2g}
\end{equation}

### 3.2 Classifications

A turbine is a device that exchange energy with a fluid using continuously flowing fluid and rotating blades. It is possible to classify these machines following many different systems:

**Fluid Flow**

- When the fluid flow goes through the machine mainly parallel to the axis of rotation, the device is termed an *axial flow turbine*. The radial component of the fluid velocity is negligible. Since there is no change in the direction of the fluid, several axial stages can be used to increase power output. A Kaplan turbine is an example of an axial flow turbine.

- When the path of the through-flow is mainly in a plane perpendicular to the rotation axis, the device is termed a *radial flow turbine*. Due to continuous change in direction, several radial stages are generally not used.

- When axial and radial flow are both present, the device is termed a *mixed flow turbine*. It combines flow and force components of both radial and axial types. A Francis turbine is an example of a mixed-flow turbine.

**Physical Action**

- *Impulse turbine* operates by accelerating and changing the flow direction through a stationary nozzle onto the rotor blade. The nozzle serves to change the incoming pressure into velocity, the enthalpy of the fluid decreases as the velocity increases. Pressure and enthalpy drop over the rotor blades is minimal.
• **Reaction turbine** operates by reacting to the flow through aerofoil shaped rotor and stator blades. In this type of turbines two effects cause the energy transfer from the flow to the mechanical energy on the turbine shaft:
  
  – Firstly, it follows from a drop in pressure from inlet to outlet of the runner. This is denoted as the reaction part of the energy conversion.
  
  – Secondly, the changes in the directions of the flow velocity vectors; the directions of the flow velocity vectors through the runner blade channels transfer impulse forces.

The velocity of the fluid through the sets of blades increases slightly (as with a nozzle) as it passes from rotor to stator and vice versa. The velocity of the fluid then decreases again once it has passed between the gaps. Pressure and enthalpy consistently decrease through the sets of blades.

### 3.3 Types of Water Turbines

Turbine selection is based on the available water head and on the available flow rate.

![Figure 17 - Comparison between Impulse and Reaction turbine configuration](image)

![Figure 18 - Rotors of different types of water turbines](image)

Hydraulic head is usually divided into three categories: low, medium, and high head. In general, impulse turbines are used for high head sites, and reaction turbines are used for low head sites.
Typical range of head are ($H$ in meters):

- Water wheel: $0.2 < H < 4$
- Kaplan turbine: $10 < H < 70$
- Francis turbine: $40 < H < 600$
- Turgo turbine: $50 < H < 250$
- Pelton turbine: $50 < H < 1300$

A brief exposition of the most used water turbines is presented in the following pages.

- **Pelton Turbine** is a tangential flow impulse turbine. This type of turbine is useful to develop power using hydraulic energy in the case of high head available. This type of turbine consists in 4 major components: collector (manifold), injector, vanes and housing. Each component is playing a significant role for better performance of machine. The energy available at the inlet of the turbine is only kinetic energy, because the pressure at the inlet and outlet is atmospheric pressure.

  ![Pelton Wheel Turbine](image)

  *Figure 19 - Pelton Wheel Turbine*

  The nozzle increases the kinetic energy of the water flowing through the penstock. At the outlet of the nozzle, the water comes out in the form of a jet and strikes the vanes of the runner. The velocity of the jet at the inlet is:

  \[
  V_{in} = C_v \sqrt{2gH_n} \tag{3.34}
  \]

  Where $C_v$ is the velocity coefficient and $H_n$ is the net head available.

- **The Francis turbine** is used primarily for medium heads and large flows applications. They are an inward-flow reaction turbine that combines radial and axial flow concepts. This turbine was developed by James B. Francis in Lowell, Massachusetts in the 1800s:
“With the canal system completed, Francis next turned his attention to turbines. Originally, the mills had used waterwheels or breast-wheels that rotated when filled with water. These types of wheels could achieve a 65% efficiency rate. One such problem with these wheels was backwater, which prevented the wheel from turning. Studying the Boyden turbine Francis was able to redesign it to increase efficiency. Constructing turbines as “sideways water wheels,” Francis was able to achieve an astounding 88 percent efficiency rate. After further experimenting, Francis developed the mixed flow reaction turbine, which later became an American standard. Twenty-two of the “Francis turbines” reside in Hoover Dam to this day.”

Their hydraulic characteristics result in relatively high-speed compact units, right up to the largest capacities. This, along with their high efficiency, has made them the most widely used turbine in the world. Francis type units cover a head range from 40 to 600 m, and their connected generator output power varies from just a few kilowatts up to hundreds megawatt. A Francis turbine consists of the following main parts: Spiral casing, guide or stay vanes, runner blades and the draft tube.

- The Kaplan turbine is a propeller-type water turbine that has adjustable blades. It was developed in 1913 by Austrian professor Viktor Kaplan, who combined automatically adjusted propeller blades with automatically adjusted wicket gates to achieve efficiency over a wide range of flow and water level. Therefore, the Kaplan turbine was an evolution of the Francis turbine. The head ranges for the Kaplan turbine are from 10 to 70 metres and the output power from 5 to 200 MW.

The Kaplan turbine is a flow reaction turbine, which means that the working fluid changes pressure as it moves through the turbine and gives up its energy. The design combines feature of radial and axial turbines. Because the propeller blades are rotated on high-pressure hydraulic oil bearings are the critical element of Kaplan design, it is essential to maintain a positive seal to prevent emission of oil into the waterway. Discharge of oil into rivers is not desirable because of the waste of resources and resulting ecological damage.

Variable geometry of the wicket gate and turbine blades allow efficient operation for a range of flow conditions. Kaplan turbine efficiencies are typically over 90%, but may be lower in very low head applications.

---

12 “The Lowell Hydraulic Experiments” 1855, Lowell Notes
3.4 Similarity Law

Dimensional analysis is a method for reducing the number and complexity of experimental variables that affect a given physical phenomenon. There are a lot of advantages of dimensional analysis and similitude in fluid mechanics.

The laws are derived from Buckingham-Pi theory. The Buckingham Pi Theorem puts the ‘method of dimensions’ first proposed by Lord Rayleigh in his book “The Theory of Sound” (1877) on a solid theoretical basis; because of that it is credited to E. Buckingham (1914). The application fields of this theorem are various. Dimensional analysis is a means of simplifying a physical problem to reduce the number of relevant variables. This analysis is particularly useful for:

- Presenting and interpreting experimental data;
- Physical modelling;
- Establishing the relative importance of particular physical phenomena;
- Checking equations.

In the turbomachinery studies about the affinity analysis play a crucial role. Sure enough, with the application of dimensional analysis and similarity or affinity laws, the number of experiments can be significantly reduced. Applying the affinity law the performance of the turbine prototype can be predicted from tests made with scaled model.

Mainly there are three similarity conditions:

- **Geometric similarity** - The model, usually scaled, is the same shape as the prototype. Therefore, the linear dimension ratio of two geometrically similar turbines are equal.

- **Kinematic similarity** - It means similarity of velocities. Fluid flow of both the model and real application must undergo similar time rates of change motions; thus, fluid streamlines are similar. Two turbines that work under kinematic similarity have the same velocity triangles.

- **Dynamic similarity** – Geometric similarity and force similarity. Therefore, the ratios between different forces in real scale prototype must be the same in model. Hence, the model scale and the real scale operate under equal conditions and efficiencies.

Thus, the comparison of two similar turbines imply:

- All the linear dimensions will be in the same ratio.
- The angles of the velocity triangles will be the same. Thus, the velocity triangles will be geometrically similar and all velocities will be in the same ratio.
Now we want to obtain a general expression to compare the behaviour of two similar turbines.

First, the discharge is proportional to the cross section multiplied by the mean velocity. Then the ratio between the model discharge $Q$ and the real turbine discharge $Q^*$ will be:

$$\frac{Q^*}{Q} = \frac{V^* D^*^2}{V D^2} \quad (3.35)$$

Therefore, because the velocity is proportional to the square root of the net head:

$$\frac{Q^*}{Q} = \frac{D^*^2}{D^2} \sqrt{\frac{H^*}{H}} \quad (3.36)$$

Then, from the similarity of the velocity triangles:

$$\frac{n^*}{n} = \frac{D}{D^*} \sqrt{\frac{H^*}{H}} \quad (3.37)$$

Therefore, under similarity condition, suppose the efficiency constant, the power:

$$\frac{P^*}{P} = \frac{Q^* H^*}{Q H} = \frac{D^*^2}{D^2} \left(\frac{H^*}{H}\right)^{3/2} \quad (3.38)$$

From the last equations, we obtain following relation:

$$\frac{n^*}{n} = \left(\frac{Q}{Q^*}\right)^{1/2} \left(\frac{H^*}{H}\right)^{3/4} = \left(\frac{P}{P^*}\right)^{1/2} \left(\frac{H^*}{H}\right)^{5/4} \quad (3.39)$$

Where $n$ is the rotational speed (rpm), $Q$ is the volumetric flow rate ($m^3/s$), $P$ is the net power produced (W), $H$ is the head at which the turbine works (m).

In these relation the terms with the star (*) corresponds to the values of the real scale turbine; the terms without the star are the values for the model.
4. Hydro-mechanical behavior

In general, there are wide ranges of material degradation mechanisms; they can be classified into three basic categories: mechanical, chemical and thermal actions.

Most of the damages in water turbines are due to mechanical degradation. In particular, there are two main types of mechanical degradation:

- Cavitation phenomenon;
- Abrasive and Erosive mechanism;

4.1 Cavitation Phenomenon

Cavitation is an important problem in hydraulic machines that may cause damage and negatively affects their performance. Cavitation occurs when the liquid pressure at a given location is reduced to the vapour pressure of the liquid. When this occurs, vapour bubbles form. The drop in pressure is caused by pushing a liquid quicker than it can react, leaving behind an area of low pressure often as a bubble of gas. These vapour bubbles collapse when they reach the blade inside the machines, which is under high pressure. These pressures are so high that they cause pitting of the blades’ metal and consequently decrease the life and efficiency of the turbomachinery.

For this reason, cavitation may be an important design consideration and it never should happen. In order to avoid the problem of cavitation, certain types and qualities of materials are chosen for the different components exposed to cavitation.

During the working of a turbine, vacuum is created at the inlet section or more rarely at the outlet section. In some situations, the vacuum may become excessive, and the cavitation occurs. Reaction turbines, basically Francis and Kaplan turbines, suitable for medium and low head hydropower sites, often suffer from cavitation phenomenon. Especially in the management of the small hydropower plants, the cavitation is an important factor; that is, to achieve higher efficiency of hydro turbines with time it is important that cavitation never occurs. When cavitation arises, turbines show declined performance after few years of operation, as they get severely damaged due to pitting of the metallic surface of the blades.

Figure 21 - Cavitation damage on the propeller's blades
When the pressure of the liquid reaches a low enough level, it vaporizes. To avoid this phenomenon, it must be:

\[ p > p_v \]  \hspace{1cm} (4.1)

Where \( p \) is the pressure of the fluid in the inlet or outlet section of the water turbine; \( p_v \) is the vapour pressure.

However, deep analyses of the turbomachinery, show that the point where the minimum pressure is reached is not exactly at the inlet or outlet section; in the inlet section it occurs at the blades’ suction surface, in the outlet section it occurs at the blades’ pressure surface of the rotor. If \( \Delta p \) is the reduction in pressure:

\[ p - \Delta p > p_v \]  \hspace{1cm} (4.2)

or

\[ p - p_v > \Delta p \]  \hspace{1cm} (4.3)

Defining the reduction as:

\[ \Delta p = \lambda \rho \frac{w^2}{2} \]  \hspace{1cm} (4.4)

Where \( \lambda \) is a constant; \( \rho \) is the fluid’s density and \( w \) is the relative velocity of the fluid. Hence:

\[ \frac{p - p_v}{\rho} > \lambda \frac{w^2}{2} \]  \hspace{1cm} (4.5)

Adding to both side of equation the kinetic term \( V^2/2 \), we obtain:

\[ \frac{p - p_v}{\rho} + \frac{V^2}{2} > \lambda \frac{w^2}{2} + \frac{V^2}{2} \]  \hspace{1cm} (4.6)

Set:

\[ gh_0 = \frac{V^2}{2} + \lambda \frac{w^2}{2} \]  \hspace{1cm} (4.7)

Where \( h_0 \) represents the head needed to accelerate the flow until the velocity \( V \).

Then:

\[ \frac{p - p_v}{g \rho} + \frac{V^2}{2g} > h_0 \]  \hspace{1cm} (4.8)
The left hand term is the **Net Positive Suction Head (NPSH)**:

\[
NPSH = \frac{p}{\gamma} + \frac{V^2}{2g} - \frac{p_v}{\gamma}
\]  

(4.9)

with

- \(\frac{p}{\gamma} + \frac{V^2}{2g}\): Total head on the suction side near the turbine impeller: inlet with \(p_{in}\) and \(V_{in}\) or outlet with \(p_{out}\) and \(V_{out}\);

- \(\frac{p_v}{\gamma}\): Liquid vapour pressure head.

There are actually two values of NPSH of interest:

- **Required NPSH (NPSH\textsubscript{R})**: This is the value that must be maintained, or exceeded, so cavitation will not occur. It is the minimum NPSH required by the turbine in order to prevent cavitation for safe and reliable operation of the turbine. The required NPSH\textsubscript{R} for a particular turbine is in general determined experimentally by the turbine manufacturer and a part of the documentation of the turbine.

- **Available NPSH (NPSH\textsubscript{A})**: This value represents the value that actually occurs for the particular system. This value is \(h_0\).

Hence, for proper turbine operation, must be:

\[
NPSH_A \geq NPSH_R
\]

(4.10)

Moreover, the Net Positive Suction Head for a water turbine can also be expressed as:

\[
NPSH = H_c - z
\]

(4.11)

Where \(z\) is the turbine vertical height (m) and \(H_c\) is the vacuum head (m). The vacuum head, at which cavitation will commence, is defined by the equation:

\[
H_c = \frac{p_{atm} - p_v}{\rho g}
\]

(4.12)

The Thomas parameter \(\sigma_{th}\) relates NPSH to the turbine head. The Thomas parameter \(\sigma_{th}\) is defined as:

\[
\sigma_{th} = \frac{NPSH}{H_c}
\]

(4.13)

where \(H_t\) is the turbine head.
To avoid the cavitation:

\[
\sigma_{th} > \sigma_{crit}
\]  

The values of the critical Thoma coefficient are obtained experimentally.

### 4.2 Abrasive and Erosive wear

The problem of abrasive and erosive wear in hydroelectric plants is really common. The dynamic action of sediment flowing along with water that impact against a solid surface cause micro holes into the blades. Therefore, this action is the root cause of sediment erosion in turbine components. Even if minor abrasion may take place in certain parts of hydro turbines, erosion is the main cause of the damage.

- **Abrasive wear** is the loss of material by the passage of hard particles over a surface. This wear occurs whenever a solid object is loaded against particles of a material that have equal or greater hardness. The abrasive wear involves processes such as micro cutting, fatigue, grain detachment and brittle fracture.

- **Erosive wear** is caused by the impact of solid and liquid particles on a surface. Erosive wear can resemble abrasive wear when hard solid particles of microscopically size are the eroding agent, the angle of impingement is low and the impingement speed is of the order of 100 m/s. For particle of microscopically size and an impingement speed of the order of 100 m/s, wear at the high impingement angles proceeds by a combination of plastic deformation and fatigue or by cracking for brittle materials.
Computational fluid dynamics (CFD) methods are employed to further understand the mechanics of hydro-abrasive erosion and, in particular, to design erosion-resistant hydraulic profiles. This typically implies lower specific speeds than normal for the given head (hence lower relative velocities), coupled with fewer jets in the case of Pelton turbines and longer, less sharply contoured blades for Francis turbines. The computerized methods can predict the region of maximum stress and then it is possible to refining the hydraulic design to mitigate the problem.

However, they are incapable of accommodating all the independent variables involves in hydro-abrasive erosion of any particular turbine. For this reason, one has to resort to semi-empirical methods based upon the myriad of data from operating turbines and from accelerated wear laboratory tests.

The simplest semi-empirical method was proposed by Zu-Yan. The formulation involved one factor: $H*C$

Where $H$ is the net head of the turbine in meters and $C$ is the average annual particle concentration in g/l of all particles with a diameter bigger than 50 µm. The ranges for hydro-abrasive erosion damage risk are therefore: $H*C \geq 7$ Severe; $0,7 < H*C < 7$ Moderate; $H*C \leq 0,7$ Negligible.

It is also possible to slightly modify this factor to obtain a more refined solution. Proposed by Nozaki as an extension of the Zu-Yan approach is the modified particle concentration factor $C$. He proposed to assume that the concentration factor as the product of the annual average particle concentration in g/l and modifying coefficients related to the variables of particle size, hardness, shape, and runner material.\(^\text{13}\)

\[ PE = P \times a \times k_1 \times k_2 \times k_3 \] (4.15)

Where \( PE \) is the modified suspension concentration (g/l), \( P \) is the measured suspended concentration (g/l), the factors \( a \), \( k_1 \), \( k_2 \), and \( k_3 \) depend on the type and geometry of the particles and type of runner material.

The following figure shows the final value of \( PE \) is used in curves of \( PE \) against turbine net head to predict times between maintenance.

*Figure 24 - Modify suspended concentration against turbine net head*
5. CFD Modelling

Computational fluid dynamics, CFD, is an increasingly important part of many development processes. Fluid flow is such an integral part to so many different projects and applications that it must be understood and optimized to improve these applications. Often the main focus of a CFD simulation is how the flow affects other process and application parameters.

Computational Fluid Dynamics provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of:

- Mathematical modelling (partial differential equations);
- Numerical methods (discretization and solution techniques);
- Software tools (solvers, pre- and post-processing utilities).

Therefore, CFD enables scientists and engineers to perform numerical experiments in a virtual flow laboratory and gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques. The following figure shows the mainly difference between the traditional studies and the CFD analysis:

<table>
<thead>
<tr>
<th>EXPERIMENTS</th>
<th>SIMULATIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quantitative description of flow phenomena using measurements</td>
<td>Quantitative prediction of flow phenomena using CFD software</td>
</tr>
<tr>
<td>• for one quantity at a time</td>
<td>• for all desired quantities</td>
</tr>
<tr>
<td>• at a limited number of points and time instants</td>
<td>• with high resolution in space and time</td>
</tr>
<tr>
<td>• for a laboratory-scale model</td>
<td>• for the actual flow domain</td>
</tr>
<tr>
<td>• for a limited range of problems and operating conditions</td>
<td>• for virtually any problem and realistic operating conditions</td>
</tr>
<tr>
<td>Error sources: measurement errors, flow disturbances by the probes</td>
<td>Error sources: modeling, discretization, iteration, implementation</td>
</tr>
</tbody>
</table>

*Figure 25 - Experimental Studies against CFD Simulations*

In a turbomachinery analysis, CFD modelling plays a really important role. In order to optimize the production of energy in a hydropower plant, it is essential that the behaviour of the propeller is well known. Therefore, the analysis of the flow through the turbine is the first step to maximize the efficiency of the turbine, hence the production of energy in the plant.

There are several software to effectuate Computational Fluid Dynamics simulations. The one that I have used is COMSOL Multiphysics®. COMSOL Multiphysics provides a complete and deep analysis of multiphysics models. This software solves the equation of the fluid flow with finite elements approximation.
5.1 Introduction

The system I have studied consists on a tubular propeller that works in a pressurized pipe. I have analysed and compared a five blade turbine (5BT) and a three blade turbine (3BT). Finally, I analyse the solution of two 5BT in series.

To perform the computational fluid dynamic analysis, I have chosen from COMSOL the Rotating machinery, Turbulent flow, $k$-$\varepsilon$ physic model. This approach is based on approximate model, then assuming this model imply the following assumptions:

- Ideal lossless turbine;
- Ignoring the elasticity of the conduit system;
- Incompressible flow;
- Turbulence in equilibrium in boundary layers;
- Assuming the water properties at 293.15 K;

These simplifications make the CFD model slightly different from the real system, but anyway it is suitable for an accurate study of the interaction between hydraulic machines and fluids. However, these imply that the model only reflect part of the real situations and as such could have a limited application.

5.2 Model Description

5.2.1 Physics

Rotating Machinery, Turbulent Flow, $k$-$\varepsilon$ interfaces are used for the modelling of flow where one or more of the domains rotate in a periodic motion. This model is usually used for mixer devices and propellers.

The mathematical model used to solve this problem is the Reynolds Averaged Navier-Stokes equations (or RANS equations). With this model the governing equations are solved in ensemble-averaged form, including appropriated models for the effect of turbulence. The objective of the turbulence models for the RANS equations is to compute the Reynolds stresses, which can be done by three main categories of RANS-based turbulence models:

- Linear eddy viscosity models
  - Algebraic models;
  - One equation models;
  - Two equation models;
- Nonlinear eddy viscosity models
  - Explicit nonlinear constitutive relation;
  - V2-f models;

- Reynolds stress model (RSM)

Specifically, this problem is solved with the k-ε model. k-ε model is a two equations linear eddy viscosity model. This model is focused on the mechanisms that affect the turbulent kinetic energy (TKE).

The assumption that this model imply is that the turbulent viscosity is isotropic, or in other words, the ratio between Reynolds stress and mean rate of deformations is the same in all directions.

In particular, the k-ε model solves for two dependent variables:

- the turbulent kinetic energy, \( k \), which describes the energy in the turbulence;
- the dissipation rate of turbulence energy, \( \varepsilon \), which describe the dissipation of the kinetic turbulence.

The two transport equation solved by the software are:

- For the turbulent kinetic energy \( k \):
  \[
  \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \varepsilon - Y_m + S_k \tag{5.1}
  \]

- For the dissipation rate \( \varepsilon \):
  \[
  \frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{C_1 \varepsilon}{k} (P_k + C_3 \sigma P_b) - C_2 \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{5.2}
  \]

Where:

\( u \) is the velocity component in the \( i \) direction

\( \mu_t \) represents the eddy viscosity: \( \mu_t = \rho C \mu \frac{k^2}{\varepsilon} \)

Where \( C_\mu \) is a constant

\( P_k \) is defined as \( P_k = \mu_t S^2 \)
Where $S$ is the modulus of the mean rate of strain tensor: $S = \sqrt{2 S_{ij} S_{ji}}$

$P_b$ introduces the effect of buoyancy: $P_b = \beta \ g_i \ \frac{\mu_t}{Pr_t} \ \frac{\partial T}{\partial x_i}$

Where $Pr_t$ is the turbulent Prandtl Number, for standard model the value of $Pr_t$ is assumed to be 0.85; $g$ is the component of gravitational vector in the $i$-direction; $\beta$ is the coefficient of thermal expansion: $\beta = -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right)_p$

Model constants:

$C_{1\varepsilon}=1.44; \ C_{2\varepsilon}=1.92; \ C_{\mu}=0.09; \ C_{3\varepsilon}=-0.33; \ \sigma_{\varepsilon}=1.0; \ \sigma_{\mu}=1.3$

The simulations were made using a Time-dependent Solver. This study type generates equations for transient (time-dependent) simulations. The Time-stepping I have chosen is the BDF (Backward differentiation formula) method.

The BDF methods is a numerical method for an ordinary differential equation (ODE), it generates an approximate solution step-by-step in discrete increments across the interval of integration. In fact, it produces a discrete sample of approximate values of the solution function. Specifically, BDFs are formulas that give an approximation to a derivative of a variable at a time $t_n$ in terms of its function values $y(t)$ at $t_n$ and earlier time-steps (hence the "backward" term).

### 5.2.2 Geometry

The CAD tools in COMSOL Multiphysics provide many possibilities to create geometries. Geometry modelling can be in 1D, 2D, and 3D. However, to create an articulate structure with irregular shapes such as a bulb turbine, it is also possible to import the geometry from a CAD software (AutoCAD or SolidWorks for instance).

In the simulations I am going to present in the next pages, the geometry of the bulb turbine is imported from the software Autodesk AutoCAD. I have designed the whole structure starting from the geometry of experimental analysis. Hence, the measurements and details of the geometry of the model are the same of the experimental system in order to obtain theoretical results that can fit the experimental ones.

The HYLOW Project (2008-2012) was a research project funded by the European Commission’s Framework 7 Program with the aim of developing novel hydropower converters for very low heads and high flow rates. This types of propeller are mainly used in mini and micro hydropower plant, even if their application for micro schemes is still underway.

The characterization of the system’s geometry is the following:
The whole system used in the laboratory for the experimental analysis makes use of a lot of parts. The main parts are the turbine's upstream bulb and runner, and the pipe. Moreover, as showed in the next figure, in the system are installed:

- Two reductions DN100/DN85, one upstream and one downstream;
- Four pressure transducers, one upstream from the runner, two immediately upstream and downstream of the runner and a fourth downstream of the curve;
- A torque sensor is installed just after the runner of the propeller, outside the pipe;
- The generator is connected to the rotating axis of the turbine.
Regardless of all the parts, in order to analyse the system with the software I need to model the propeller and the pipe with the proper shape. Through COMSOL Multiphysics I simulate the flux of the water inside the pipe; thus I have to create the domains where the water goes through.

To design the system geometry on AutoCAD I have started with the propeller. The following figures show both three and five blades turbines:

![Figure 28 – 3BT CAD Model: 3D Visualization](image)
![Figure 29 – 5BT CAD Model: 3D Visualization](image)

Next, I have created the pipe. I have chosen to leave a gap between the propeller’s blades and the internal wall of the pipe of 2 mm, thus the diameter of the pipe is 90 mm.

Afterwards, to perform the CAD model with the CFD study, I have needed to create an extra cylinder to define the rotating domain inside the pipe. This cylinder goes around the propeller, as the following figure shows:
The insides of the propeller and the shaft are empty. Moreover, I have neglected to model the thin layer of fluid that is rotating around the shaft. I have chosen to do that because the mesh of that thin layer consist in a huge number of tiny elements, thus the computational time increases excessively. However, neglecting that does not influence much the final results that much.

Hence, the whole system is composed of 2 different domains:

- Domain 1 consists of the water inside the pipe. To simplify the analysis, I create only the inside wall of the pipe; then there is not the thickness of the pipe. For this reason, the whole Domain 1 is filled up with water.

- Domain 2 consists of the small cylinder empty of the propeller. This is the rotating domain. The propeller volume is empty.

The domains imported in COMSOL are showed in the Figure 31.

5.2.3 Material

COMSOL Multiphysics has predefined material database available to build models. From the Built-In database I have chosen the material I needed. Since I have chosen to having no pipe thickness and to having empty space for the propeller, the only material I need is water. The two domains are filled up with water.

Afterwards, in the Material Contents section I have entered the following water’s properties evaluated at 293,15 K:

- Density: \( \rho = 1000 \text{ kg/m}^3 \)
Dynamic Viscosity: $\mu = 0.001 \text{ Pa}\cdot\text{s}$

### 5.2.4 Boundary Conditions

Under *Rotating Machinery, Turbulent Flow*, $k$-$\varepsilon$ section I have added the boundary condition to solve the problem. Basically, I have imposed:

- The rotation velocity for the rotating domain (Domain 2);
- The velocity in the inlet section;
- The pressure in the outlet section;
- Flow continuity between the two domains.

Moreover, the software itself imposes the fluid properties, no slip condition on the walls, the rotating walls and the initial values for velocity field and pressure equal to zero.

### 5.2.5 Mesh

The Mesh features enable the discretization of the geometry model into small units of simple shapes. For the 3D geometry the mesh generator discretizes the domains into tetrahedral, hexahedral, prism, or pyramid elements. The boundaries in the geometry are discretized into triangular or quadrilateral boundary elements.

I have chosen to discretize the domains into tetrahedral elements. The size of the elements is different between the two domains. In the rotating domain, the elements are smaller, because of the turbulence create by the turbine, it is necessary to compute the solution in a bigger number of nodes.

The following tables show the numbers of elements for the three systems:

<table>
<thead>
<tr>
<th></th>
<th>3BT</th>
<th>5BT</th>
<th>Series-5BT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain 1</td>
<td>0.025</td>
<td>0.015</td>
<td>0.025</td>
</tr>
<tr>
<td>Domain 2</td>
<td>0.01</td>
<td>0.005</td>
<td>0.02</td>
</tr>
<tr>
<td>Maximum element size</td>
<td>0.025</td>
<td>0.002</td>
<td>0.01</td>
</tr>
<tr>
<td>Minimum element size</td>
<td>0.015</td>
<td>0.003</td>
<td>0.02</td>
</tr>
<tr>
<td>Maximum element growth rate</td>
<td>1.15</td>
<td>1.45</td>
<td>1.25</td>
</tr>
<tr>
<td>Resolution of curvature</td>
<td>0.6</td>
<td>0.6</td>
<td>0.7</td>
</tr>
<tr>
<td>Resolution of narrow regions</td>
<td>0.8</td>
<td>0.7</td>
<td>0.7</td>
</tr>
</tbody>
</table>

*Table 1 - Mesh details*
6. Analyses and Results

In the following section the results of the CFD simulations are showed. The main parameters of the models are: the flow rate $Q \, [\text{m}^3/\text{h}]$; the pressure head $H \, [\text{mWc}]$; the rotation speed $n \, [\text{rpm}]$.

Fixing the flow rate and the pressure, I have made several tests for each turbine configuration changing the rotation speed, specifically:

- **3BT** - I have run this model using five different rotation speeds: 250 rpm, 500 rpm, 750 rpm, 1000 rpm, 1500 rpm.

- **5BT** – Since from the analysis of the experimental results it is clear that the maximum efficiency is reached around 750 rpm and 1000 rpm, I have made the simulations for these two values of rotation speed.

- **Series-5BT** – I have obtained the simulation for four simulations: for a flow rate of 16 $\text{m}^3/\text{h}$ and 32 $\text{m}^3/\text{h}$; in both cases I analysed the model for two rotational speed, 750 rpm and 1000 rpm.

In this chapter, I have compared the results of the simulations of 3BT, 5BT and Series-5BT with a rotation speed of 750 rpm. However, I have attached all the other simulations in the Appendix.

6.1 3BT and 5BT Results

6.1.1 Velocity Field

One of the main aims in fluid dynamics is to find the velocity field $\vec{v}$ to describe the flow in a given domain. The basic equations of fluid dynamic flow used to obtain the solution are:

- **Continuity equation** (or conservation of the mass). The continuity equation is obtained by equalizing the rate of decrease of mass and the total rate of mass flux. Hence, the general equation that describes the conservation of the mass for a specific domain is:

  \[
  \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0
  \]  

  \text{(6.1)}

  Specifically, if the flow is incompressible, then the density does not change in time. Thus, the equation become:

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

  \text{(6.2)}

- **The Conservation of Momentum**, for incompressible flow, is defined as:

  \[
  \rho \frac{D\vec{v}}{Dt} = \rho \vec{g} - \nabla p + \mu \Delta \vec{v}
  \]  

  \text{(6.3)}
Where $\frac{D\vec{v}}{Dt}$ is the total derivate of the velocity vector and $\nabla$ is the Laplacian operator.

The following picture (Figure 33) shows the velocity field $\vec{v} = (u, v, w)$ in the 3BT and 5BT models, for the rotational speed of 750 rpm.

![Figure 32 - Velocity Field: a) 3BT and b) 5BT](image)

In both figures, we can clearly see that the velocity is the highest around the turbine, which is expected, because of the rotation speed. It is also evident, that the velocity magnitude between the two model is quite different. Even if the initial values, the inlet velocity and the rotation speed of the turbine are the same in both simulations, the velocity field is influenced by the turbine geometry.

![Figure 33 - Velocity Field in a perpendicular section: 3BT and 5BT](image)

Analysing now a section that cross the blades, it is possible to see where the maximum velocity occurs. It is clear from Figure 34, the points where the maximum velocity is reached are different for the two models.
First of all, the reason why the blades of the 3 blade turbine seem so thick in this cross section, is due to their inclination; the inclination of the blade in 3BT model is smaller than the 5BT model, respectively 19.9° and 31.4°.

In the 3BT model, maximum velocity is reached between two consecutive blades. This happens because of the geometry of the model; there is little space, as the Figure 35 shows. Instead, because in the section of the 5BT there are not free space, the maximum velocity occurs just on the blade surface.

In the 3BT model velocity reaches a lower value than the 5BT model. The average velocity magnitude in the rotating domain are: 0.85 m/s in the 5BT model and 0.75 m/s in the 3BT, while the maximum magnitude velocity values are: 3.47 m/s in the 5BT model and 3.23 m/s in the 3BT.

### 6.1.2 Shear-Stress Velocity

The shear-stress velocity (also known as the friction velocity), $u_*$, of a flow is defined by the relation:

$$ u_* = \frac{\tau}{\sqrt{\rho}} $$

(6.4)

Where $\tau$ is the wall shear stress and $\rho$ the fluid density. The friction velocity is often used as a scaling parameter for the fluctuating component of velocity in turbulent flows.
The shear-stress velocity characterises the turbulent strength and the thickness of turbulent boundary layer.

The higher shear-stress velocity is obviously reached around the blades of the turbine. It means that in the turbine blades the shear stress $\tau$ reaches higher values.

In the 3BT model the shear stress velocity is around 2.7 m/s, but in the 5BT model it reaches 4.6 m/s. The different values are due to the different shape of the blade. In the 3BT there is more space between two blades, then the shear stress developed in that region is much lower than for the 5 blade turbine.

### 6.1.3 Vorticity Field

Turbulent flows are rotational; therefore, they have non-zero vorticity. Mechanisms such as the stretching of three-dimensional vortices play a key role in turbulence. Vorticity is a vector field that gives a microscopic measure of the rotation at any point in the fluid. The vorticity field is related to the amount of local angular rate of rotation in a fluid flow. More precisely, the vorticity is a field $\vec{\omega}$, defined as the curl of the flow velocity $\vec{v}$ vector. In Cartesian coordinate, it is defined as:

$$\vec{\omega} = \nabla \times \vec{v} = \left( \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \times (v_x, v_y, v_z)$$

Where $\nabla$ is the del operator.

Hence the vorticity express how the velocity vector changes when one moves by an infinitesimal distance in a direction perpendicular to it.
Therefore, it is obvious that the rotation of the turbine inside the pipe causes a vorticity field. The higher the rotational speed, the more intense is the vorticity field. Figure 37 shows the vorticity field for the two models. Even if the rotation speed is the same, 750 rpm, because of the different number of blades and their different shape the vorticity developed inside the pipe appears different.

![Figure 36 - Vorticity Field: a) 3BT and b) 5BT](image)

Since in the 3BT the turbulence is higher than in the 5BT, the losses will be considerable and the efficiency will not be high.

Exactly as we expected, the region where vorticity is higher, the red ones, are where the velocity reaches the higher values.

![Figure 37 - Vorticity Field in a perpendicular section: 3BT and 5BT](image)
6.1.4 Turbulent Kinetic Energy and Dissipation rate

The turbulent kinetic energy, \( k \), is the kinetic energy per unit mass of the turbulent fluctuation in a turbulent flow; therefore, the SI unit of \( k \) is \( J/kg = m^2/s^2 \).

Since in turbulent flow the velocity is defined as the sum of a constant value and a fluctuating value:

\[
\mathbf{u}_i(t) = \bar{\mathbf{u}}_i + \mathbf{u}'_i(t)
\]  
(6.6)

Hence, the turbulent kinetic energy is generally defined as:

\[
k = \frac{1}{2} (u'^2 + v'^2 + w'^2)
\]  
(6.7)

Where \( u' \), \( v' \) and \( w' \) are the components of the fluctuating velocity, \( \mathbf{u}_i(t) \).

The turbulent intensity is:

\[
T = \frac{\sqrt{\frac{2}{3} k}}{U_{\text{ref}}}
\]  
(6.8)

Turbulent regime flows are dissipative. Kinetic energy gets converted into heat due to viscous shear stresses. Turbulence dissipation rate is the rate at which turbulence kinetic energy is converted into thermal internal energy.

*Figure 38 - Velocity in turbulent regime flow*

*Figure 39 - Turbulent Kinetic Energy: a) 3BT and b) 5BT*
I made four simulations for the series turbine:

- For a flow rate of 16 m$^3$/h and for an outlet pressure of 0,34 bar, for both rotational speed of 750 rpm and 1000 rpm;
- For a flow rate of 32 m$^3$/h and for an outlet pressure of 0,20 bar, for both rotational speed of 750 rpm and 1000 rpm;

In this section I compare the results for the rotational speed of 750 rpm; the other results are showed in the Appendix.
It is evident from Figure 42, that velocity reaches different values in the two models. Because of the discharge increase, since the cross section of the pipe is constant, then velocity increases \( \left( V = \frac{Q}{A} \right) \). Therefore, respectively the second picture shows a velocity field with higher values, especially around the blades of the turbine.

Instead, about the direction of the velocity vector, we can verify that it is the same in both simulations. The stream lines follow the same direction in both pictures. Regarding the vorticity field, the same consideration is true.

![Figure 42 - Series-SBT: Vorticity Field: a) Q=16 m³/h; b) Q=32 m³/h](image)

Regarding the vorticity field, even if the values of the field are different for the two models, the stream lines follow the same directions. As for the single turbine, the intensity of turbulence is much more stronger around the turbine blades but also around the shaft and in the bend of the pipe.

Figure 44 shows the TKE and the corresponding dissipation rate.

![Figure 43 - Series-SBT: TKE: a) Q=16 m³/h; b) Q=32 m³/h](image)
Since the turbulent kinetic energy is computed as $\frac{1}{2}(u'^2 + v'^2 + w'^2)$, as we expected the highest values of the TKE are reached in the same points where the velocity has the maximum values; and properly, in the second simulations turbulent kinetic energy developed around the blade, is much higher than the first simulation. Moreover, because of the dissipation rate represents the dissipation of the kinetic energy, Figure 45 shows that the trend is the same of the TKE.

It is interesting to notice, that the second turbine develop much more kinetic energy therefore the dissipation rate is higher too. This behaviour can depend on the turbulence of the flow. Since in the first turbine the turbulence regime is smoother than in the second turbine, this increase a lot the kinetic energy and dissipation rate in the second turbine. Moreover, this can explain why the efficiency of the second turbine is a little bit lower than the first one.

**6.3 Efficiency Analysis**

To compute the efficiency, I have used the following equations:

$$\eta = \frac{P_{mec}}{P_h}$$

(6.9)

Where $P_{mec}$ is the mechanical power and $P_h$ is the hydraulic power.

I have computed the mechanical power with the following equation:

$$P_{mec} = T \omega$$

(6.10)

Where T is the torque that I obtained from the simulation and $\omega$ is the rotational speed (rad/sec).
Since the hydraulic power $P_h$ is:

\[ P_h = \rho g Q H \]  \hspace{1cm} (6.11)

Hence, I have all the data to computed the efficiency.

*Figure 46* shows the trend of the efficiency against the rotation speed. Afterwards the corresponding values on the tables.

\begin{table}[h]
\centering
\begin{tabular}{|c|c|c|c|c|c|}
\hline
Rotation Speed [rpm] & H [mWc] & Torque [Nm] & $P_m$ [W] & $P_h$ [W] & $\eta$ [%] \\
\hline
250 & 0.24 & 0.139 & 3.65 & 9.35 & 35.27 \\
500 & 0.24 & 0.069 & 3.62 & 10.35 & 35.02 \\
750 & 0.23 & 0.038 & 2.97 & 9.91 & 29.92 \\
\hline
\end{tabular}
\caption{3BT CFD results: Efficiency}
\end{table}

\begin{table}[h]
\centering
\begin{tabular}{|c|c|c|c|c|c|}
\hline
Rotation Speed [rpm] & H [mWc] & Torque [Nm] & $P_m$ [W] & $P_h$ [W] & $\eta$ [%] \\
\hline
750 & 0.39 & 0.144 & 11.30 & 16.81 & 67.22 \\
1000 & 0.38 & 0.098 & 10.26 & 16.39 & 62.62 \\
1250 & 0.32 & 0.060 & 6.32 & 13.80 & 45.53 \\
1500 & 0.31 & 0.041 & 4.27 & 13.37 & 31.96 \\
\hline
\end{tabular}
\caption{5BT CFD results: Efficiency}
\end{table}
From the trend, it is clear that the best efficiency point for the two configurations are different. When the turbine works with 3 blades, the maximum efficiency is reached for low rotation speed; for 250 rpm the efficiency is maximum, then it decreases.

As we expected, from the experimental results, for the 5 blades turbine the best efficiency point is for a rotation speed of 750 rpm.

Regarding the series turbine, I have used the same relations of single turbine. The following tables show the results obtained:

<table>
<thead>
<tr>
<th>Q=16 m³/h</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Turbine 1</strong></td>
</tr>
<tr>
<td>Rotational Speed [rpm]</td>
</tr>
<tr>
<td>750</td>
</tr>
<tr>
<td>1000</td>
</tr>
<tr>
<td><strong>Turbine 2</strong></td>
</tr>
<tr>
<td>Rotational Speed [rpm]</td>
</tr>
<tr>
<td>750</td>
</tr>
<tr>
<td>1000</td>
</tr>
</tbody>
</table>

*Table 4 - Series-SBT efficiency for Q=16 m³/h*

<table>
<thead>
<tr>
<th>Q=32 m³/h</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Turbine 1</strong></td>
</tr>
<tr>
<td>Rotation Speed [rpm]</td>
</tr>
<tr>
<td>750</td>
</tr>
<tr>
<td>1000</td>
</tr>
<tr>
<td><strong>Turbine 2</strong></td>
</tr>
<tr>
<td>Rotation Speed [rpm]</td>
</tr>
<tr>
<td>750</td>
</tr>
<tr>
<td>1000</td>
</tr>
</tbody>
</table>

*Table 5 - Series-SBT efficiency for Q=32 m³/h*

These tables shows that even with two turbines inside the pipe, it is possible to maintain the efficiency quite high. Therefore, theoretically, with the same one fluid flow it is possible to develop much more power than only using one turbine.
6.4 Experimental and CFD Results

The HYLOW project developed the experimental analysis of the 5 blade tubular turbine. The results of the performance assessment are presented in Figure 32, where we can analyse the efficiency trends.

![Figure 32](image1.png)

**Figure 32 – Efficiency and Power trend (Source: Experimental characterization of a five blade tubular propeller for inline installation, Samora I. 2016)**

For a fixed rotation speed, efficiency increases rapidly with the increase of the discharge until a maximum is reached and then it slowly starts to decrease. Nevertheless, the mechanical power always increases with the increase of the discharge, showing that the augmentation of the head compensates the decreasing of efficiency.

I analysed the simulations for these values of flow rate and rotational speed, in order to be able to compare the results of the simulations with the experimental results. Clearly, the efficiency computed using the simulations is a little bit higher than the experimental one. Because of the assumptions of the CFD model, this result is reasonable. However, the trends of the CFD results are the same of the experimental charts.

For both results, the best efficiency point (BEP) corresponds to a flow rate of 16 m³/h and a rotation speed of 750 rpm. The best efficiency is around 64% for the experimental results, while it is 67,2% for the simulations results. Table 6 shows the mechanical power developed by three similar turbines. Clearly, since the BEP is for 750 rpm, the higher power is developed around this rotational speed.

<table>
<thead>
<tr>
<th>Rotational Speed [rpm]</th>
<th>Mechanical Power [W]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>16 m³/h</td>
</tr>
<tr>
<td>750</td>
<td>16,81</td>
</tr>
<tr>
<td>1000</td>
<td>16,39</td>
</tr>
<tr>
<td>1250</td>
<td>13,8</td>
</tr>
<tr>
<td>1500</td>
<td>13,37</td>
</tr>
</tbody>
</table>

*Table 6 – SBT CFD results: Mechanical Power for different flow rate*
Finally, a brief overview about the economical point of view. These type of turbines are under research, so the following formulas are only a rough approximation based on the laboratory manufacture.

It is analysed a recent research of a hydro-power plant installation scheme with a pump as turbine (PAT)\textsuperscript{14}, and the following results are obtained.

The cost of each turbine is proportional to the power developed at the best efficiency point. For this example the power developed if around 10 kW:

\[
C_{\text{turbine}} (\€) = 150 \left( \frac{\€}{\text{kW}} \right) P_{\text{BEP}} (\text{kW})
\]  

while the cost of the generator is:

\[
C_{\text{generator}} (\€) = 105 \left( \frac{\€}{\text{kW}} \right) P_{\text{max}} (\text{kW})
\]  

which is proportional to the maximum 5BT power.

\textsuperscript{14} “PAT Design Strategy for Energy Recovery in Water Distribution Networks by Electrical Regulation”, Armando Carravetta, Giuseppe del Giudice, Oreste Fecarotta and Helena M. Ramos, 2013
6.5 Affinity Law Analysis

Clearly from tables 2 and 3, the mechanical power developed from the turbine is quite low. Mainly this happen because the diameter of the turbine it is not big in size and the flow rate is modest. To expand this results to different turbine size, it is possible to apply the affinity laws, for different values of turbine diameters and flow rate.

I have made this analysis only for the 5 blades turbine, because it exhibits the best behaviour in comparison with 3 blades turbine. Starting from the equation of chapter 3.4, and considering the case of constant rotation speed, I have applied the following equation to obtain the new pressure head, $H$, and mechanical power $P_m$:

\[ Q = Q^* \frac{N}{N^*} \left( \frac{D}{D^*} \right)^3 \]  \hfill (6.14)  

\[ H = H^* \left( \frac{N}{N^*} \right)^2 \left( \frac{D}{D^*} \right)^2 \] \hfill (6.15)  

\[ P_m = P_m^* \left( \frac{N}{N^*} \right)^3 \left( \frac{D}{D^*} \right)^5 \] \hfill (6.16)

Exactly as in chapter 3.4 in these relation the terms with the star (*) corresponds to the values of the real scale turbine, that now they are the values that we want to obtain, and the terms without the star are the values of the model.

Therefore, using these relations, I analyse the changing of mechanical power for a 5 blades turbine with two different diameter and flow rate. *Table 7* and *Table 8* show the results that I have obtained.

**Case 1:** Diameter of 200 cm and flow rate of 37,27 m$^3$/h

<table>
<thead>
<tr>
<th>Rotation Speed [rpm]</th>
<th>$H$ [mWc]</th>
<th>Torque [Nm]</th>
<th>$P_m$ [W]</th>
<th>$P_h$ [W]</th>
</tr>
</thead>
<tbody>
<tr>
<td>750</td>
<td>2,16</td>
<td>1,876</td>
<td>147,25</td>
<td>211,60</td>
</tr>
<tr>
<td>1000</td>
<td>2,10</td>
<td>1,277</td>
<td>133,66</td>
<td>206,17</td>
</tr>
<tr>
<td>1250</td>
<td>1,77</td>
<td>0,625</td>
<td>81,83</td>
<td>173,62</td>
</tr>
<tr>
<td>1500</td>
<td>1,72</td>
<td>0,354</td>
<td>55,65</td>
<td>168,19</td>
</tr>
</tbody>
</table>

*Table 7 - Affinity Analysis: Case 1*

**Case 2:** Diameter of 400 cm and flow rate of 74,54 m$^3$/h

<table>
<thead>
<tr>
<th>Rotation Speed [rpm]</th>
<th>$H$ [mWc]</th>
<th>Torque [Nm]</th>
<th>$P_m$ [W]</th>
<th>$P_h$ [W]</th>
</tr>
</thead>
<tbody>
<tr>
<td>750</td>
<td>8,64</td>
<td>15,00</td>
<td>1178,03</td>
<td>1692,79</td>
</tr>
<tr>
<td>1000</td>
<td>8,42</td>
<td>10,22</td>
<td>1069,29</td>
<td>1649,38</td>
</tr>
<tr>
<td>1250</td>
<td>7,09</td>
<td>5,01</td>
<td>654,67</td>
<td>1388,96</td>
</tr>
<tr>
<td>1500</td>
<td>6,87</td>
<td>2,83</td>
<td>445,17</td>
<td>1345,55</td>
</tr>
</tbody>
</table>

*Table 8 - Affinity Analysis: Case 2*
7 Conclusion

In conclusion, since big hydropower plants are used since many decades, they are deeply known and commonly used all over the world. Instead, the micro-hydro converters are not well known yet. However in the last years, the research about this new configurations of turbine were developed. To improve and speed up this process, CFD modelling plays an essential role.

Overall, nonetheless the simplification assumed since the difficult of the real situation, CFD simulations present a valid option to extract reasonable results capable to analyse the fluid flow behaviour itself and study the interactions between the fluid and the turbine.

The turbines for micro-hydropower applications are analysed using the CFD software COMSOL Multiphysics. The systems that present the best results are the 5 blade turbine and the 5 blade turbine in series. The results of the 5BT simulations show that the best efficiency point is reached for a flow rate of 16 m$^3$/h for a rotational speed of 750 rpm. The maximum efficiency is 67,22%.

Regarding the 5BT in series, the maximum efficiency is reached for a flow rate of 32 m$^3$/h for a rotational speed of 750 rpm. The efficiency of the turbines is 72,3% and 68,4%. This result is really outstanding; the system of series turbine could be an efficient solution for example for drinkable water or irrigation conduit where the head available is really low while the flow rate is remarkable.
8 Bibliography


ASCE Hydropower Task Committee (2007). *Civil Works for Hydroelectric Facilities: Guidelines for the Life Extension Upgrade*, American Society of Civil Engineers


Parr N.M., Charles J.A. and Walker S. (1992). *Water resources and reservoir engineering*, University of Stirling


Glossary

**Abrasion**: Loss of material by the passage of hard particles over a surface.

**Alternating current (AC)**: Electric current that reverses direction many times per second.

**Ancillary services**: Capacity and energy services provided by power plants that are able to respond on short notice, such as hydropower plants, and are used to ensure stable electricity delivery and optimized grid reliability.

**Barrage**: Artificial obstruction built in a watercourse to increase its depth or to divert its flow.

**Biomass**: Organic matter, especially plant matter, that can be converted to fuel and is therefore regarded as potential energy source.

**Cavitation**: The phase changes that occur from pressure changes in a fluid that forms bubbles, resulting in noise or vibration in the water column. The implosion of these bubbles against a solid surface, such as a hydraulic turbine, may cause erosion, and lead to reductions in capacity and efficiency pressure.

**Control gate**: A barrier that regulates water released from a reservoir to the power generation unit.

**Direct current (DC)**: Electric current which flows in one direction.

**Draft tube**: A water conduit, which can be straight or curved depending upon the turbine installation, which maintains a column of water from the turbine outlet and the downstream water level.

**Efficiency**: A percentage obtained by dividing the actual power or energy by the theoretical power or energy. It represents how well the hydropower plant converts the potential energy of water into electrical energy.

**Erosion**: Caused by the impact of solid and liquid particles on a surface. Erosive phenomenon can resemble abrasive phenomenon when hard solid particles of microscopically size erode the surface.

**Fish ladder**: A transport structure for safe upstream fish passage around hydropower projects.

**Flow**: Volume of water, expressed as cubic feet or cubic meters per second, passing a point in a given amount of time.

**Fossil Fuel**: Any kind of combustible organic material, as oil, coal or natural gas, derived from the remains of former life.

**Friction**: It is any force that resists relative tangential motion; its direction is opposite the relative velocity.

**Geothermal Energy**: The natural heat produced inside the Earth.
**Generator:** Device that converts the rotational energy from a turbine to electrical energy.

**Head:** Vertical change in elevation, expressed in feet or meters, between the head (reservoir) water level and the tailwater (downstream) level.

**Headwater:** The water level above the powerhouse or at the upstream face of a dam.

**Hydropower:** The harnessing of flowing water—using a dam or other type of diversion structure—to create energy that can be captured via a turbine to generate electricity.

**Impoundment:** A body of water formed by damming a river or stream, commonly known as a reservoir.

**Micro hydro:** Hydropower projects that generate up to 100 kilowatts.

**Penstock:** A closed conduit or pipe for conducting water to the powerhouse.

**Pitting:** Corrosion that is localized in a certain part of the surface. It caused cavities or holes in the material. Pitting is considered to be more dangerous than uniform corrosion damage because it is more difficult to detect, predict and design against.

**Power house:** The structure that houses generators and turbines

**Pumped storage:** A type of hydropower that works like a battery, pumping water from a lower reservoir to an upper reservoir for storage and later generation.

**Reservoir:** See impoundment.

**Rotor:** The rotating member of the hydraulic machine. The device has a number of blades radiating from central hub that is rotated to produce thrust to produce energy. The stator is usually enveloped the rotor (see Stator).

**Runner:** See Rotor.

**Scroll case:** A spiral-shaped steel intake guiding the flow into the wicket gates located just prior to the turbine.

**Shear Stress:** The external force acting on a body or surface parallel to the slope or plane in which it lies; the stress tending to produce shear.

**Small hydro:** Hydropower projects that generate 10 MW or less of power.

**Solar Energy:** Energy derived from the sun’s radiation. Solar energy can be converted into thermal or electrical energy through solar panel.
**Spillway:** A structure used to provide the release of flows from a dam into a downstream area.

**Stator:** The part of a machine that remains fixed with respect to rotating part (see Rotor).

**Tailrace:** The channel that carries water away from a dam.

**Tailwater:** The water downstream of the powerhouse or dam.

**Transformer:** Device that takes power from the generator and converts it to higher-voltage current.

**Turbine:** A machine that produces continuous power in which a wheel or rotor revolves by a fast-moving flow of water.

**Vapour pressure:** The pressure exerted by the vapour which is in equilibrium with liquid phase in a closed system at a specific temperature.

**Viscosity:** Formally, viscosity is the ratio of the shear stress to the velocity gradient. Viscosity is the quantity that describes a fluid's resistance to flow.

**Wicket gates:** Adjustable elements that control the flow of water to the turbine.

**Wind Energy:** Power derived from the wind. It is used to generate electricity through some wind turbines.
Appendix

3BT – TEST1

Simulation Data:

- Volume flow rate - 16 m³/h
- Rotation speed – 250 rpm (36,2 rad/sec)

Velocity Field

*Figure 48 - 3BT: Velocity Field (250 rpm)*

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 49 - 3BT: Velocity Field, Slice representations (250 rpm)
Shear-Stress Velocity Field

Figure 50 - 3BT: Shear-Stress Velocity (250 rpm)

Vorticity Field

Figure 51 - 3BT: Vorticity Field (250 rpm)
Turbulent Kinetic Energy, $k$

Figure 52 - 3BT: Turbulent Kinetic Energy (250 rpm)

Dissipation Rate, $\varepsilon$

Figure 53 - 3BT: Dissipation Rate (250 rpm)
3BT – TEST2

Simulation Data:

- Volume flow rate - 16 m³/h
- Rotation speed – 500 rpm (52.3 rad/sec)

**Velocity Field**

![Velocity Field Image](image)

*Figure 54 - 3BT: Velocity Field (500 rpm)*

The following pictures show the velocity field, using the slice representation, for six different time steps.
Figure 55 - 3BT: Velocity Field, Slice representations (500 rpm)
Shear-Stress Velocity Field

Figure 56 - 3BT: Shear-Stress Velocity (500 rpm)

Vorticity Field

Figure 57 - 3BT: Vorticity Field (500 rpm)
**Turbulent Kinetic Energy, k**

*Figure 58 - 3BT: Turbulent Kinetic Energy (500 rpm)*

**Dissipation Rate, ε**

*Figure 59 - 3BT: Dissipation Rate (500 rpm)*
3BT – TEST3

Simulation Data:

- Volume flow rate - 16 m³/h
- Rotation speed – 750 rpm (78.5 rad/sec)

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 60 - 3BT: Velocity Field, Slice representations (750 rpm)
3BT – TEST4

Simulation Data:

- Volume flow rate - 16 m$^3$/h
- Rotation speed – 1000 rpm (104.7 rad/sec)

Velocity Field

*Figure 61 - 3BT: Velocity Field (1000 rpm)*

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 62 - 3BT: Velocity Field, Slice representations (1000 rpm)
Shear-Stress Velocity Field

Figure 63 - 3BT: Shear-Stress Velocity (1000 rpm)

Vorticity Field

Figure 64 - 3BT: Vorticity Field (1000 rpm)
**Turbulent Kinetic Energy, k**

*Figure 65 - 3BT: Turbulent Kinetic Energy (1000rpm)*

**Dissipation Rate, ε**

*Figure 66 - 3BT: Turbulent Dissipation Rate (1000rpm)*
**3BT – TEST5**

Simulation Data:

- Volume flow rate - 16 m³/h
- Rotation speed – 1500 rpm (157.1 rad/sec)

**Velocity Field**

*Figure 67 - 3BT: Velocity Field (1500 rpm)*

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 68 - 3BT: Velocity Field, Slice representations (1500 rpm)
Shear-Stress Velocity Field

Figure 69 - 3BT: Shear-Stress Velocity (1500 rpm)

Vorticity Field

Figure 70 - 3BT: Vorticity Field (1500 rpm)
Turbulent Kinetic Energy, $k$

Figure 71 - 3BT: Turbulent Kinetic Energy (1500rpm)

Dissipation Rate, $\epsilon$

Figure 72 - 3BT: Turbulent Dissipation Rate (1500rpm)
**5BT – TEST1**

Simulation Data:

- Volume flow rate - 16 m³/h
- Rotation speed – 750 rpm (78.5 rad/sec)

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 73 - 5BT: Velocity Field, Slice representations (750 rpm)
5BT – TEST2

Simulation Data:

- Volume flow rate - 16 m³/h
- Rotation speed – 1000 rpm (104.7 rad/sec)

**Velocity Field**

*Figure 74 - 5BT: Velocity Field (1000 rpm)*

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 75 - SBT: Velocity Field, Slice Representation (1000 rpm)
Shear-Stress Velocity Field

Figure 76 - SBT: Shear-Stress Velocity (1000 rpm)

Vorticity Field

Figure 77 - SBT: Vorticity Field (1000 rpm)
Turbulent Kinetic Energy, $k$

Figure 78 - SBT: Turbulent Kinetic Energy (1000 rpm)

Dissipation Rate, $\varepsilon$

Figure 79 - SBT: Dissipation Rate (1000 rpm)
Series-5BT – TEST1

Simulation Data:

- Volume flow rate - 16 m$^3$/h
- Rotation speed – 750 rpm (78,5 rad/sec)

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 80 - Series-SBT: Velocity Field, Slice representations (16 m³/h - 750 rpm)
Series-5BT – TEST2

Simulation Data:

- Volume flow rate - 16 m³/h
- Rotation speed – 1000 rpm (104.7 rad/sec)

**Velocity Field**

*Figure 81 - Series-5BT: Velocity Field (16 m³/h – 1000 rpm)*

The following pictures show the velocity field, using the slice representation, for six different time steps.
Figure 82 - Series-SBT: Velocity Field, Slice representations (16 m³/h - 1000 rpm)
Shear-Stress Velocity Field

Figure 83 - Series-SBT: Shear-Stress Velocity (16 m³/h – 1000 rpm)

Vorticity Field

Figure 84 - Series-SBT: Vorticity Field (16 m³/s – 1000 rpm)
Turbulent Kinetic Energy, $k$

Figure 85 - Series-5BT: Turbulent Kinetic Energy (16 m$^3$/h – 1000 rpm)

Dissipation Rate, $\varepsilon$

Figure 86 - Series-5BT: Dissipation Rate (16 m$^3$/h – 1000 rpm)
Series-5BT – TEST3

Simulation Data:

- Volume flow rate - 32 m³/h
- Rotation speed – 750 rpm (104.7 rad/sec)

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 87 - Series-SBT: Velocity Field, Slice representations (32 m³/s - 750 rpm)
Series-5BT – TEST4

Simulation Data:

- Volume flow rate - 32 m³/h
- Rotation speed – 1000 rpm (104.7 rad/sec)

**Velocity Field**

*Figure 88 - Series-5BT: Velocity Field (32 m³/h – 1000 rpm)*

The following pictures show the velocity field, using the slice representation, for six different time step.
Figure 89 - Series-5BT: Velocity Field, Slice representations (32 m³/h - 1000 rpm)
Shear-Stress Velocity Field

Figure 90 - Series-SBT: Shear-stress Velocity (32 m³/s – 1000 rpm)

Vorticity Field

Figure 91 - Series-SBT: Vorticity Field (32 m³/s – 1000 rpm)
**Turbulent Kinetic Energy, $k$**

*Figure 92 - Series-SBT: Turbulent Kinetic Energy (32 m$^3$/h – 1000 rpm)*

**Dissipation Rate, $\varepsilon$**

*Figure 93 - Series-SBT: Dissipation Rate (32 m$^3$/h – 1000 rpm)*