Universitat de Girona
Escola Politècnica Superior

Treball final de master

Estudi: Màster en Enginyeria Industrial

Títol: Optimization study of the performance of an impulse-type turbine with CFD

Document: Master thesis

Alumne: Tim Ongena

Tutor: Toni Pujol
Departament: Eng. Mecànica i de la Construcció Industrial
Àrea: Mecànica de Fluids

Convocatòria (mes/any): June 2017

Foreword

This master thesis is part of my Erasmus study at the University of Girona, to complete my study 'Master of Electromechanical Engineering Technology', started at the University of Antwerp.
Special mention goes to Dr. Toni Pujol, who I would like to thank for his excellent guiding throughout the months, as well as his patience.

Table of contents

1 Introduction

Water mills have long been used to convert hydraulic energy to mechanical power. The ancient Roman culture was the first known society that widely used water wheels for this purpose (mainly devoted to grinding grain). After the industrial revolution took place in the XVIII century, and with the invention of alternating current and electrical generators, the water wheel gained another function, which consisted on producing electricity from the kinetic energy of water. Today water wheels (see Figure 1) are still being used in rural communities (in Nepal for example) to gain energy from a neighbouring water flow. The main difference from the older versions is that new designs now make use of integrated technological advances like steel blades, and therefore have a better efficiency than classical water wheels. Another advantage is that they nowadays can be utilised for more than one purpose at once [1].

As it can be seen in Fig. 1, water wheels nowadays employed in isolated communities use an horizontal axis configuration instead of the classical vertical one. The reason is that this configuration is expected to increase the efficiency of ancient devices, especially if, as it is shown in Fig. 1, a closed conduit instead of a waterfall is used for directing the upstream water. However, the installation of these new horizontal-axis water turbines follows a trial-and-error method for determining the geometrical configurations that are expected to optimize the power output. It is clear that a more systematic study about the influence of the several design parameters in the net efficiency of these turbines is needed.

Therefore, this project tries to find the optimal configuration of an impulse jet water turbine by researching the most influential parameters. We propose the application of the Taguchi method for being able to create a geometrical setup that produces a maximum power output. This statistical approach is necessary since there are six different elements that are expected to have a large influence on the transfer of the energy of the water, namely: 1) the number of blades, 2) the inclination angle of the blades, 3) the inclination angle of the water jet, 4) the height of the water jet with respect to a reference point (see further information in the next chapters), 5) the horizontal position of the nozzle and 6) the peripheral velocity of the water wheel. Each one of these six parameters may vary within a large range of values, so it is not feasible to simulate every combination. Therefore, we apply the Taguchi method to reduce the number of experiments (i.e., simulations) needed for determining an improved configuration.

The different setups will be simulated with ANSYS 17.1 academic software, where we will work in the CFD-environment ANSYS CFX. The geometry of the simulated turbine will correspond to the horizontal axis impulse type one situated in the Hydraulics Laboratory of the Department of Mechanical Engineering and Industrial Construction of the University of Girona.

Figure 1: Vertical water mill in Nepal. This picture was taken out of a published paper, reference number [1].

2. Methodology

To conduct the research successfully, we must fulfil different tasks as described below.

Firstly, we must modify the original geometry provided from a previous work in order to be able to parameterise the several parts of the turbine listed previously. These parameters form an integral part of the used method developed inside the Design Modeller software. This is the native CAD software offered by ANSYS. Thus, by working with the Design Modeller software, we will avoid any interpretation problem related to the process of importing geometries from external sources. This is how we assure that our configuration of the setup will go smoothly.

Secondly, we have the mesh. We must ensure that the number of elements does not exceed the limit of 512000 (we have used the student version of ANSYS Workbench) for all cases, including those with the maximum amount of water blades (22). Since the original design from a previous work almost met this number of elements, we must probably choose a mesh that is slightly less accurate. This could have consequences on the accuracy of the simulated results.

Thirdly, we should decide on the different setups for using the Taguchi method. We cannot choose the vertical and horizontal tube position without taking the geometry of the turbine into account. Furthermore, we should always try to make the impact of the waterjet as perpendicular as possible to the blade surface in order to gain a maximal power output.

Before starting with all the simulations required from the Taguchi method, we will carry out few simulations with a fixed setup but varying the turning velocity. This will give us an idea as to where the ideal value of the rotation turbine speed may be, so we can make a proper decision regarding the range for this parameter.

When we have met the previously described conditions we can simulate the different setups (accordingly to Taguchi method) and analyse the results. The Taguchi method claims that we can see the parameters that have the biggest influence on the efficiency of the turbine [2] and afterwards we can modify these to find an optimal result. This is done by implementing the two-step optimisation method for Taguchi designs. This means that after the first step, which is finding the imperative factors, we should adjust the level of one or more factors who considerably affect the mean but not the signal to noise ratio. If the resulting output is not ideal, we will continue the search by the trial and error approach, and use the results of the first step in the Taguchi method as guidelines for finding an optimal configuration.

Furthermore, we will include the necessary information about the geometry, mesh and setup in the annexes. This will be done to ensure an easy replication of the results and a smooth transition for new researchers on this topic in case that further investigations will be required.

3. Horizontal axis impulse type turbine

Before we can start with the different setups, we must first identify the parameters that influence the power output of the water wheel. It is probably easiest to look at the theoretical analysis [3] of the output of the turbine to find them. The values of the variables correspond to the experimental setup in the hydraulics laboratory[1].

3.1 Power output

Since we need to find the power, we need a transfer of energy in a certain period. This is done by the impact on the blades of the water coming out of the nozzle. So, we will start with the effect of the energy conversion of water. The total energy is given by:

$$E_t = E_k + E_p \tag{1}$$

where E_k is the kinetic energy of the incoming water jet and E_p is the potential energy of the waterjet where the base of the turbine acts as the reference point to calculate the height.

Dividing equation (1) by the time we get the power, which gives the following expression for the theoretical power induced by the waterjet:

$$E_t = \frac{P_t}{t} \tag{2}$$

$$P_t = P_k + P_p \tag{3}$$

With P_k and P_p the kinetic and potential power terms respectively.

The first term P_k can be calculated with the common formula for the power of a waterjet:

$$P_k = p_{dynamic}\dot{Q} \tag{4}$$

with $p_{dynamic}$ the dynamic pressure and $\dot{Q}$ the mass flow rate, being

$$P_k = \frac{1}{2}\rho_{water}v^2\dot{Q} \tag{5}$$

where ρ_{water} is the water density and v the mean velocity of the incoming flow.
Since the mass flow and water jet velocity of our laboratory turbine are equal to

$$\dot{m} = \frac{V\rho}{t} = \frac{100l*1\frac{dm}{l}*1\frac{kg}{dm^3}}{20.65s} = 4.84\,\frac{kg}{s} = 4.84\,\frac{l}{s} \tag{6}$$

$$v = \frac{\dot{Q}}{A} = \frac{\dot{Q}}{\frac{\pi}{4}*d^2} = \frac{4.84\frac{dm^3}{s}}{\frac{\pi}{4}*20.7mm} = 14.39\,\frac{m}{s} \tag{7}$$

the kinetic power of the incoming waterjet is

$$P_k = \frac{1}{2} * 1000 * 14.39^2 * 0.00484 = \ 501W \tag{8}$$

[1] The dimensions of the turbine can be found in Annex A.

Thus, the total power of the incoming jet is

$$P_t = \frac{\dot{m}v^2}{2} + \dot{m}gh \tag{9}$$

$$P_t = 501\,W + 10\,W = 511\,W \tag{10}$$

where

$$h = 0.209m$$

since *'h'* (the relative height of the nozzle with respect to the bottom of the box) has an influence – be it a small one – we can include it as a parameter for Taguchi. However, the main influence of h on the output power will not come from the potential energy term but from the position of the waterjet impact on the blade surfaces.

Since the different configuration per case will decide the resulting torque, the manner of obtaining the torque is also explained. ANSYS CFX is a finite volume method code that can calculate forces due to water impacts over any solid surface. In addition, the program can provide the torque values with respect to the X-, Y-, and Z- axis.

We will start with an analysis of the generated power based on the dynamics of systems. In a static environment, the torque, or moment, is a resulting rotational force generated by an applied force 'F' on a lever (or in our case the blades), which is situated at a certain distance 'r' from the rotational centre (of the turbine).

Thus, the formula for the torque is given by

$$T = F * r \tag{11}$$

Then, the dynamic of systems provides us with a formula to calculate the peripheral speed of the rotating part of the turbine, which is given by formula:

$$v = \omega * r \tag{12}$$

If we multiply the torque T with the angular velocity ω, we get the, simplified, formula for power

$$P[W] = T\,[Nm] * \omega\,[\frac{rad}{s}] \tag{13}$$

Eq. (13) is valid only for points situated at a distance r from the centre of rotation. In our case, this distance varies since the surface elements in which we have discretized our blades are located at different r. Therefore, ANSYS integrates a power differential over the variations of r. This is the way to get the torque via ANSYS in turbomachinery. The previous work on this topic managed to do this by defining the necessary expressions when configuring the ANSYS CFX setup. The explanation on how to do this can be found in Chapter 5.3.

When we have the result of an experiment (or simulation), the monitor will show a graph as in (Fig. 2). These are the user defined points as explained in Chapter 5.3, where the blue line represents the torque generated by the periodical impact of the waterjet on the blades (moment along the z-direction in agreement with the definition of our coordinate system). The periodicity shown in the blue data corresponds to the time interval between two consecutive blades. Because the model uses the symmetry of the water wheel, the generated torque curves show only half of the true value. Therefore, we should always double the output power to get the correct value of generated power.

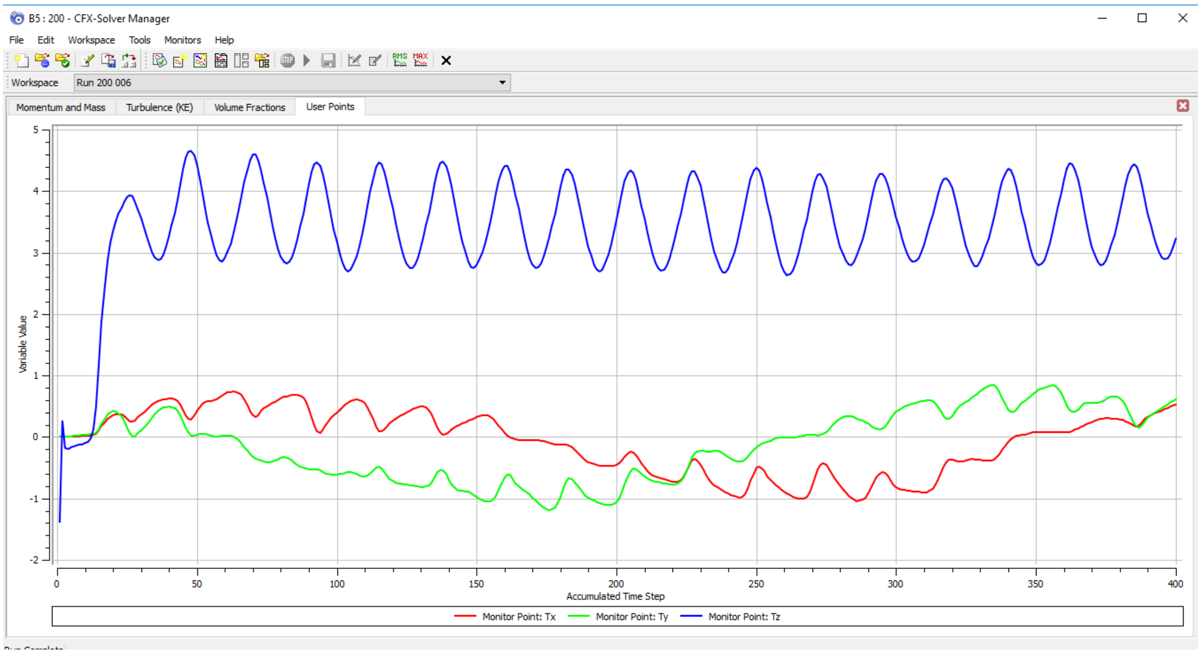


Figure 2: Example of half of the generated torque, a consequence of the symmetry, as a result of the water jet having an impact on the blades. Red and green lines represent the torque values along the X and Y directions, respectively.

To calculate the average value of the torque, based upon the Z-axis, we export the plot variables to a csv file. Afterwards, we implement these in ‘Excel’ and take the top to top average of one full cycle of the water turbine. To get the power we must multiply this result with the angular velocity used for that case, as illustrated in Table 1: Example of calculating the powerTable 1:

Table 1: Example of calculating the power

Turning velocity (rpm)	250
Turning velocity (omega; rad/s)	26.18
Torque (N m)	3.53
Power (W)	184.8

Since the angular velocity has a clear influence on the power, we did some experiments with a setup as it was presented at the start of this thesis. The parameters had the values as shown in Table 2:

Table 2: Test values for power curve in function of the peripheral velocity

Vertical tube position (mm)	172
Horizontal tube distance (mm)	73
Angle_Inlet (°)	0
Number_Of_Blades	16
Blade inclination (°)	180
Rotating speed	variable

The different power outputs can be found on Figure 3, where we observe that the turning velocity of the turbine of the turbine that produces the maximum power, is between 250 rpm and 350 rpm.

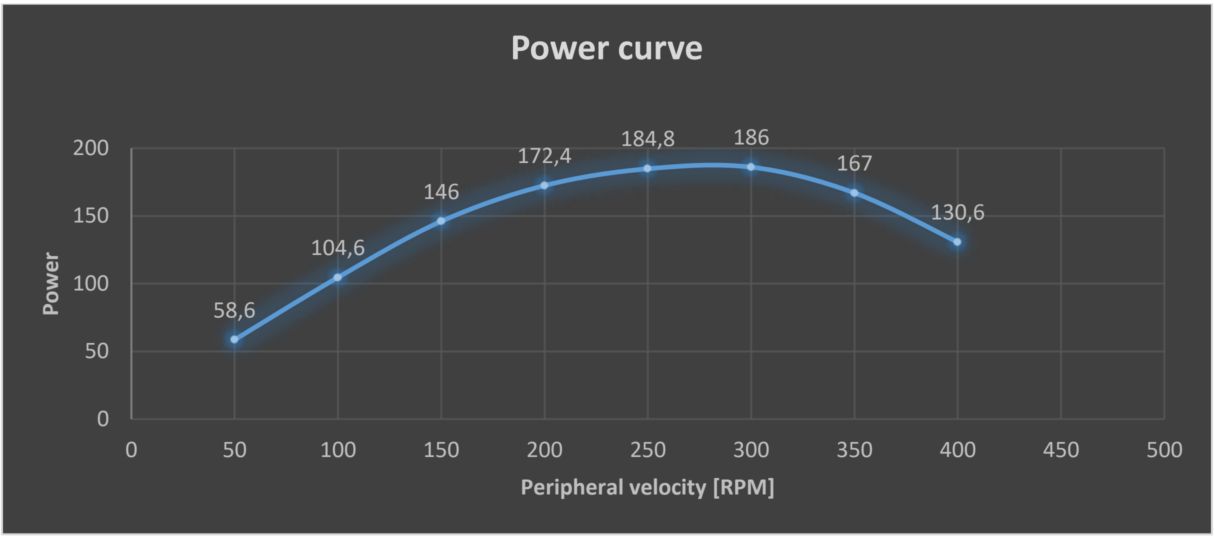


Figure 3: Power curve generated by the water turbine, where we the power output is doubled as a result of the symmetry of the model of the turbine.

Another indication of the best angular velocity are the results of a different optimisation study [4], as shown in Fig. 4. In Figure 4, the angle inlet was 0°, the blade inclination was 180°

d=20,7	n [rpm]								
h [mm]	50,00	100,00	200,00	250,00	260,00	270,00	280,00	300,00	400,00
631,00				187,30	189,82	190,33			
633,00				188,67	191,39	192,31	192,01		
634,00				188,80	191,43	192,43	191,23		
635,00			175,00	189,50	191,32	190,84		185,00	
640,00			170,34	179,00	181,86	181,05		181,00	
646,00	56,24	99,79	156,96	167,00	164,03	162,61		162,28	95,61
653,00				132,96		131,22			

Figure 4:Power output values from another research regarding the same turbine [3]

and the height of the horizontal tube with respect to the base of the turbine box was '*h'*. Note that close to the maximum power, changes of the power output less than 1% are found in a range of turning velocities as large as 20 rpms (equivalent to a 7% of the turning velocity at the maximum power).

3.2 Parameters

In this subsection, we describe in more detail the six parameters that we vary in our optimization study.

The first (height of the nozzle) and second (turning velocity) parameters can be derived from the previous paragraphs. The height of the nozzle and the turning velocity both have an impact on the power output, although one might be more important than the other.

A third factor is the horizontal position of the nozzle (see Fig. 5). This length influences the area where the waterjet has an impact on the blades as it is shown in Fig. 5 where we have depicted three possible inclined configurations of the tube. Indeed, the effect of this position on the blades also depends on the inclination angle of the tube and on the inclination angle of the blades (fourth factor).

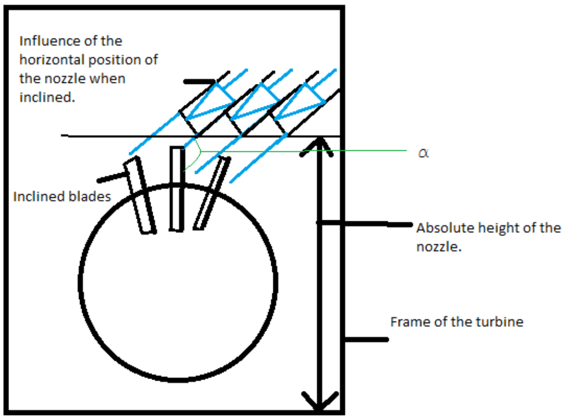


Figure 5: Horizontal position of the nozzle in combination with the absolute height of the tube.

This brings us to the next parameter (5^{th} factor): number of blades. The number of blades influences the duration of the impact from the waterflow on the blades, as shown in Table 3 below.

Table 3: Blade to blade angular distance in function of the number of blades

Number of blades	Angular distance (°)
14	25.7
16	22.5
18	20
20	18
22	16.36

The values shown in Table 3 are calculated as follows, with the case of 18 blades serving as an example.
The water wheel has a circular form, so if we have 18 blades spread evenly over the 360°, every blade touches the water flow over an angle of 20°, with $-10° < \theta < +10°$.

The last parameter (6th factor) is the inclination of the nozzle, since it also influences the angle of impact and might give way to a more perpendicular impact, which would improve the efficiency.

An overview of the Taguchi parameters is shown in Table 4.

Table 4: The identified parameters of influence

Factor	Parameters	Units
A	Rotating speed	rpm
B	Vertical tube position	mm
C	Horizontal tube distance	mm
D	Angle nozzle	Degrees
E	Blade inclination	Degrees
F	Number of blades	/

To clarify the meaning of the angle of the nozzle and the inclination of the blades, Figure 6 is included.

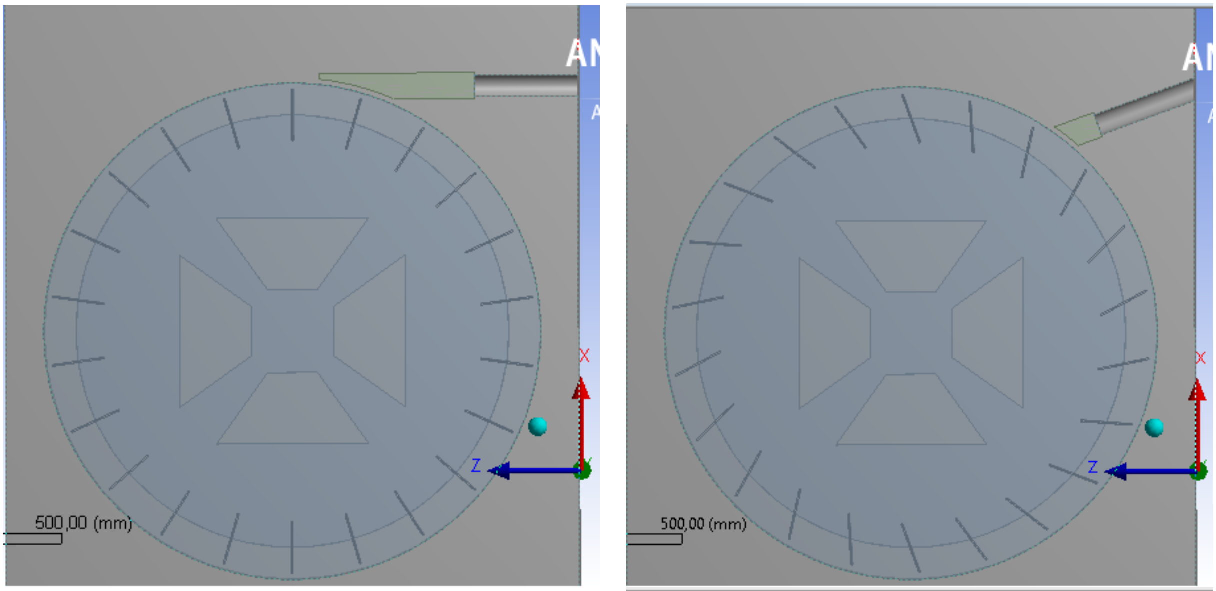


Figure 6: Left: turbine setup with a blade inclination angle of 180° and an inlet tube angle of 0°. Right: blade inclination set on 160° and the angle of the nozzle at -20°.

4 Taguchi method

During the past few decades, new optimisation methods have been rapidly created and enhanced. One of the possible methods to gain enough information about all combinations in a process is the full factorial design, founded by the English statistician Sir R.A. Fischer. This approach has a rather big disadvantage since the amount of experiments needed for it is equal to all possible combinations. It was Dr. Taguchi who used orthogonal arrays not only to control the error in experiments, but also to measure the variation from the average result [2]. An extra addition to the original method is the possibility to map the experimentation factors and their responding interactions to the correct columns of the array. This leads to a better understanding of all factors since their effects can be studied in an independent manner, without contamination from one another and this eliminates the need for carrying out all the experiments.

4.1 Orthogonal arrays

4.1.1 Orthogonality

The easiest way to describe the orthogonal arrays method would be '*being balanced and not mixed'*. In this experimental context, this translates to statistically independent relationships in every array and column. We will explain this with an example in an L_4 orthogonal array shown in Table 5.

Table 5: An orthogonal L_4 array

L_4 (2^2) orthogonal array		
	Independent variables	
Experiment number	Variable 1	Variable 2
1	1	1
2	1	2
3	2	1
4	2	2

First, we will explain the nomenclature. The 'L_x 'represents the amount of experiments needed, and depends on the number of parameters, as well as their levels, influencing the process. The array shown in Table 5 is an example of a (2x2) array, because each variable has two different levels. The general notation would be $\#levels^{\#variables}$. The orthogonality is there because both variables are included two times in the matrix in the rows as well as in the columns.

In our experimental setup, we will use the L_{25} (5^6) orthogonal array, shown in Table 6 on the next page, therefore reducing the needed experiments from 15.625 to an achievable 25. When we notice big differences in results it means that changing the level from a factor has a significant influence on the measured characteristic, which is in our case the power or efficiency of the water turbine. Because every level occurs the same number of times for every factor, any effect of the factors on each other will be cancelled out, which confirms the independency of the variables.

To use the Taguchi method to its full extent, the engineer should plan the experimental setup very carefully, since not all interaction combinations can be measured as is the case with a full factorial approach. To meet this requirement, we selected the only six parameters that can have an effect on the maximum power output, thereby reducing the possibility of doing simulations that appear to be redundant in hindsight.

4.1.2 Properties of an orthogonal array

As mentioned in the previous paragraph the Taguchi method vastly reduces the necessary amount of conducted experiments to find reliable results. This comes because of the following traits:

1. Balancing property of the orthogonal array: under the independent factors (or variables) we can find a certain set of levels, where every one of them appears an equal number of times.
2. Every value of each level must be used for the experiments.
3. The sequence of the level values must remain the same.

Following these rules leads to the experimental setup as shown in Table 6.

Table 6: The orthogonal array L_{25} setup for the experiment

L_{25} (5^6) orthogonal array						
Run	**Factor**					
	A	**B**	**C**	**D**	**E**	**F**
1	1	1	1	1	1	1
2	1	2	2	2	2	2
3	1	3	3	3	3	3
4	1	4	4	4	4	4
5	1	5	5	5	5	5
6	2	1	2	3	4	5
7	2	2	3	4	5	1
8	2	3	4	5	1	2
9	2	4	5	1	2	3
10	2	5	1	2	3	4
11	3	1	3	5	2	4
12	3	2	4	1	3	5
13	3	3	5	2	4	1
14	3	4	1	3	5	2
15	3	5	2	4	1	3
16	4	1	4	2	5	3
17	4	2	5	3	1	4
18	4	3	1	4	2	5
19	4	4	2	5	3	1
20	4	5	3	1	4	2
21	5	1	5	4	3	2
22	5	2	1	5	4	3
23	5	3	2	1	5	4
24	5	4	3	2	1	5
25	5	5	4	3	2	1

4.2 Data analysis

To get the relevant information out of the simulations, one must segregate the individual effects of independent variables. This can be done by taking the mean values of output power and efficiency for every given factor and its level. We will compare the results by introducing a new parameter, the signal to noise (S/N) ratio, which allows us to investigate the sensitivity of the different factors on the process more clearly. This ratio has long been used by engineers to retrieve the most influential parts regarding power output in various environments. The signal stands for the desired real value, whilst the noise refers to the undesired factors in measured values. The bigger the difference between the extreme values, the higher the quality of the performance will be.

Since the S/N ratio has multiple categories to seek the best result for an experiment, we must choose the correct one. Our goal is to maximise the power output (=y), so the method used should be the 'Larger is better' approach. The thus adopted formula is then given by

$$^{S}/_{N} = -10\log\left(\frac{1}{y^2}\right) \tag{14}$$

If we combine the power and efficiency in the same formula, we get

$$^{S}/_{N} = -10\log\left(\frac{1}{y_p^2}+\frac{1}{y_\eta^2}\right) \tag{15}$$

as a result.

To analyse the data, we will do the following. First, we receive the power per run as an output from the simulations. Then, we will calculate the S/N ratio from every different run. Afterwards we must segregate the results to their respective variables, which is what we described earlier. As a result, we will get a graph where the mean S/N ratio will be plotted on the y-axis and the level at the x-axis as we can see in Fig. 7 on the next page. On this example, it is easy to see the parameter who has the biggest impact on the process (E), followed by A and so on. It is feasible that by trying to idealise these two aspects whilst keeping the S/N ratio the same, the process will be optimised. This is also what we will attempt.

The Taguchi method has been applied with success in several cases of optimization of parameters related with engineering processes and by means of numerical simulations in ANSYS [4].

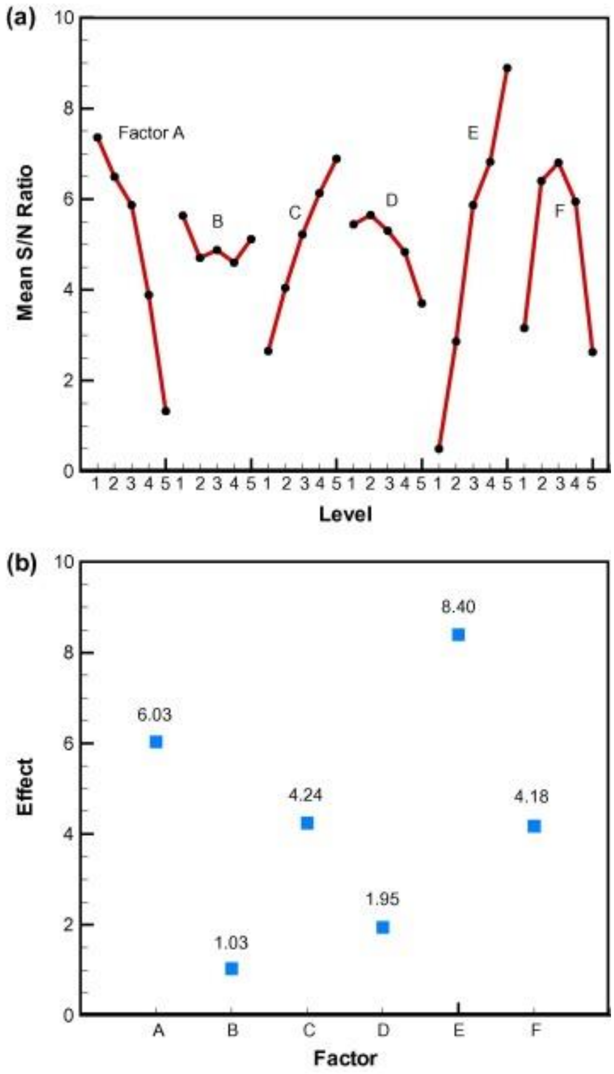


Figure 7: Above: an example of a plotted S/N ratio for each factor (A to F) as a function of the levels (1 to 5) in a L25 orthogonal array case. Below: range of variation for each factor. This picture was taken out of a published paper, reference no. [5]

4.3 Cases

As explained above it is imperative to make the correct choices regarding the ranges of values for the parameters if we want to achieve good simulation results. This means that the choices for the settings need to be realistic in relation to each other, as well as to the geometry.
Table 7 shows the values for every parameter:

Table 7: The level values

Factor	Parameter	Unit	Level				
			1	2	3	4	5
A	Rotating speed	rpm	200	237.5	275	312.5	350
B	Vertical tube position	mm	146.95	158.2	169.45	180.7	191.95
C	Horizontal tube distance	mm	73	79.75	86.5	93.25	100
D	Angle inlet	Degrees	0	-5	-10	-15	-20
E	Blade inclination	Degrees	160	165	170	175	180
F	Number of blades (-1)	/	14	16	18	20	22

Note 1: The vertical tube position is referenced to a coordinate system at the relative height of 464.65 mm with respect to the bottom of the turbine. This has been done because of the modifications necessary for the parametrisation.

If we insert the values of Table 7 in Table 2, we get Table 9 as a result, which will be the settings for every run and can be found on the next page. In the results, there will be two extra columns. One with the resulting power following each run, the other one will represent the S/N ratio.

Table 8 Shows the different values for the time steps.

Table 8: Time steps for the peripheral speeds

Rotating speed (rpm)	200	237.5	275	312.5	350
Time step value (s)	0.0008333	0.0007018	0.0006061	0.0005333	0.0004762

The manner of calculating the time steps is explained in chapter 5.3. The shown values of the time step Δt correspond with a constant change of angular position of the water wheel equal to 1 degree (independently on their respective rotating velocity N (in rpm)):

$$\Delta t = \frac{1^{\text{º}}}{\frac{N\left(\frac{rev}{min}\right)360\left(\frac{^\circ}{rev}\right)}{60\ \left(\frac{s}{min}\right)}} = \frac{1}{6N}$$

Table 9: Taguchi configuration of a $L_{25}\,5^6$ orthogonal array

Run	Factor					
	A	**B**	**C**	**D**	**E**	**F**
1	200	146.95	73	0	160	14
2	200	158.2	79.75	-5	165	16
3	200	169.45	86.5	-10	170	18
4	200	180.7	93.25	-15	175	20
5	200	191.95	100	-20	180	22
6	237.5	146.95	79.75	-10	175	22
7	237.5	158.2	86.5	-15	180	14
8	237.5	169.45	93.25	-20	160	16
9	237.5	180.7	100	0	165	18
10	237.5	191.95	73	-5	170	20
11	275	146.95	86.5	-20	165	20
12	275	158.2	93.25	0	170	22
13	275	169.45	100	-5	175	14
14	275	180.7	73	-10	180	16
15	275	191.95	79.75	-15	160	18
16	312.5	146.95	93.25	-5	180	18
17	312.5	158.2	100	-10	160	20
18	312.5	169.45	73	-15	165	22
19	312.5	180.7	79.75	-20	170	14
20	312.5	191.95	86.5	0	175	16
21	350	146.95	100	-15	170	16
22	350	158.2	73	-20	175	18
23	350	169.45	79.75	0	180	20
24	350	180.7	86.5	-5	160	22
25	350	191.95	93.25	-10	165	14

5 CFD Model

The configuration of the model that should be optimised can be found in the hydraulics laboratory in the University of Girona and it is shown in Figure 8. Before we can implement the Taguchi parameters we should modify the available setup to our demands. More detailed information is included in the annexes. The different cases and simulations will be done using the ANSYS R17.1 software, whilst utilising the Fluid Flow CFX software code. These are performed on computers with OS Windows 7, processor Intel 5 Core 4 Quad Core.

Figure 8: Laboratory water wheel. In the upper right of the left figure you can see the tube from which the water exits. Note the straight blades used in the system as well as the horizontal axis in the right picture.

5.1 Geometry and parameterisation

The original geometry was made in the Design Modeler software from ANSYS by another master student, Sven Cornelis [4]. However, in order to be able to use the method of Taguchi in an effective way, with all its different setups, it is useful to parameterise the parts that have an influence on the power output. Since we have now identified the important factors, we can modify the geometry of the model.

The modifications[2] are not radical. The main thing is to redefine the origin of the sketches and to make sure that the axis of rotation has the correct direction in order to rotate correctly. After this comes the parameterisation, which is simply done by selecting the white box in front of the details view in ANSYS Design Modeler software. As to highlight the fact that something is parameterised, a blue 'D' appears in the box as shown in Figure 9.

[2] More detailed information about the geometry can be found in B 1 Geometry.

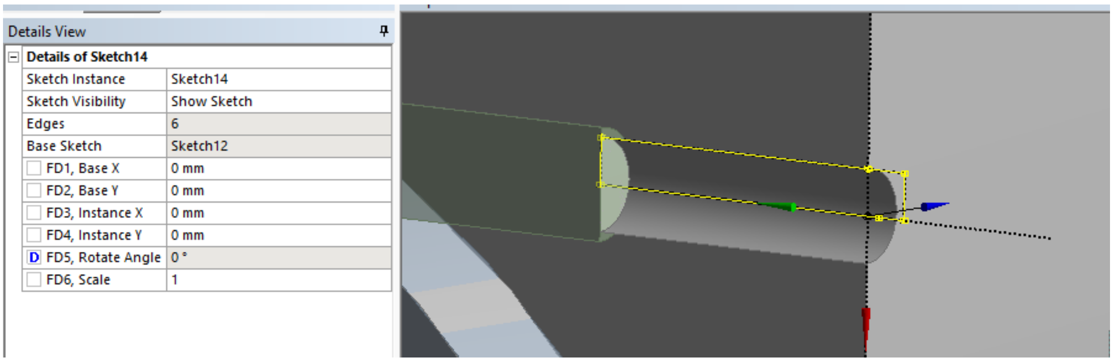


Figure 9: Parameterisation of the inclination of the nozzle

We have had to modify a sketch on one occasion. This is done in order to define the inclination angle of the blades because ANSYS Design Modeler software does not allow applying an inclination to a full sketch. Consequently, we must only draw half of the height of the blade and applying an angle constraint to it. After this, the sketch is completed and the blade angle can then be parameterised. To get the multiple blades, we make use of the command 'pattern', with the one sketch of the blade acting as the base in the ANSYS Design Modeler software. This however results in a small anomaly, since the pattern command does not include the element that is chosen as a base, which means that we must always select one blade less in the parameter area than we want to have on the rotating part, as shown in Figure 10.

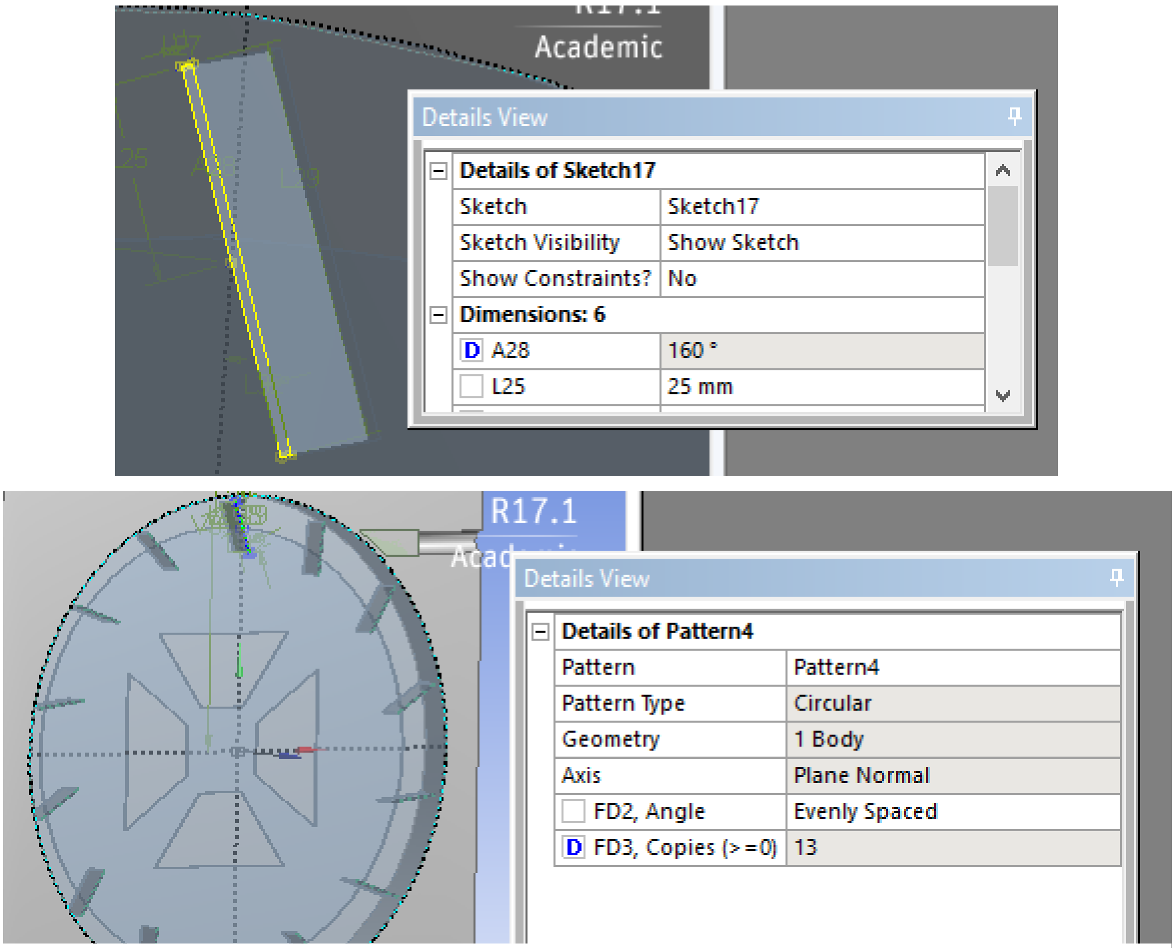


Figure 10: Parameterised inclination of the blades and their pattern. The actual number of blades is 14 in this case. One original blade and 13 added by the pattern command.

5.2 Mesh

Because the method of Taguchi requires ranges for the levels, the original mesh must be adjusted to accommodate the 512.000 elements limit. The reason for this is that the accuracy of the mesh in specific regions, in this case the blades, are very important in order to achieve accurate simulation results. Here we will have a maximum of 22 blades instead of the original 16 (as the turbine in the laboratory). Therefore extra blades require a vast number of extra elements if we want to work with the same accuracy of the simulations, as shown in Figure 11.

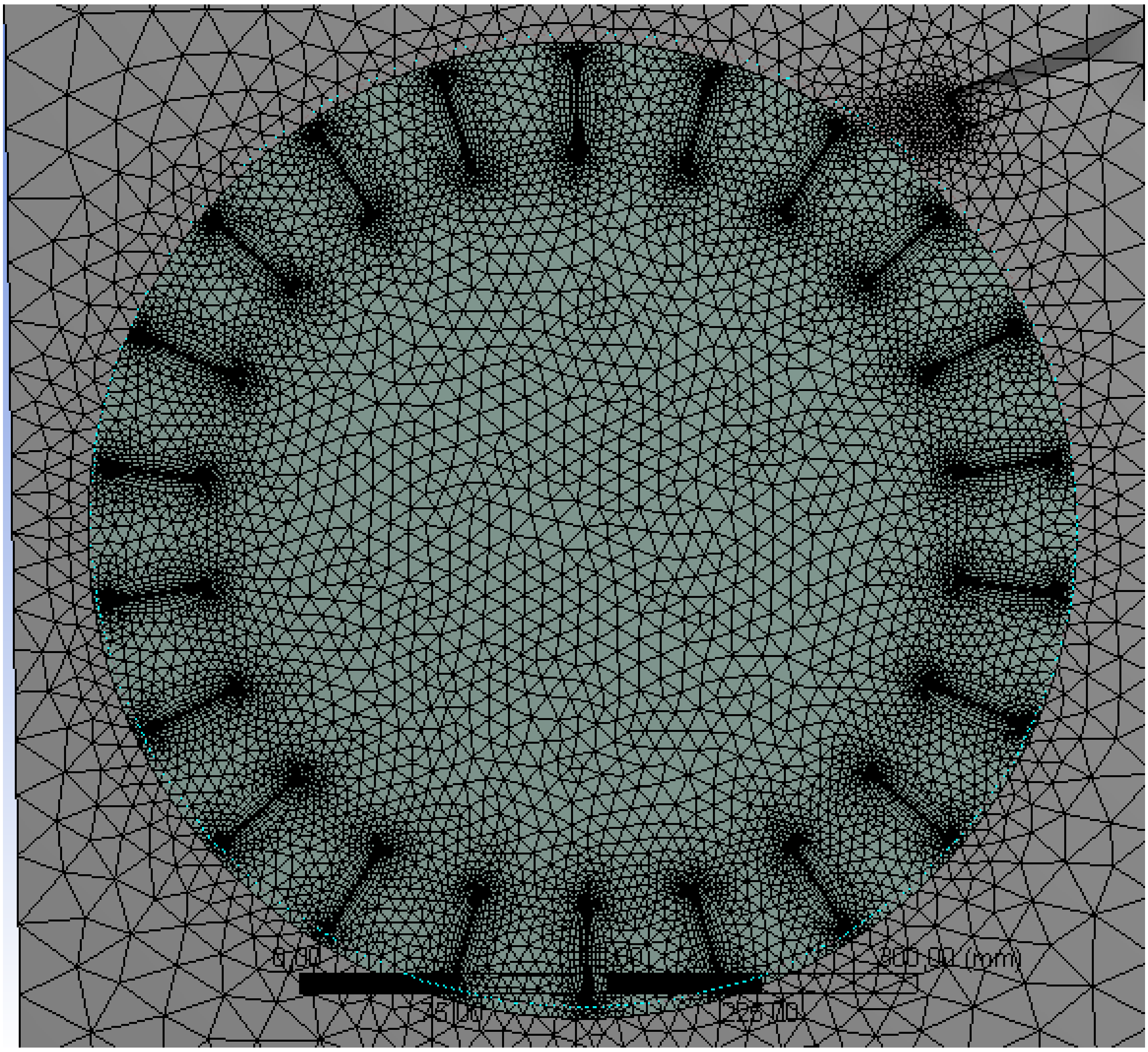


Figure 11: Geometry with 22 blades and inclined nozzle, mesh from run 5.

This leads to 511.775 elements which is just below the limit of 512.000 so we can use this mesh for simulations with the student version of the software. In order to be able to stay under the limit for the number of elements we have to modify some of the original meshing settings. One of them is the growth rate of the elements near the blades. Initially, this is set at 1.2 to ensure a very accurate calculation around the blades. We change this to 1.3 so every element is now a maximum of 30% bigger than the previous one. Furthermore, we select a high smoothing compared to the medium from before, to improve the mesh quality, as shown in Figure 12.

The quality of the mesh can be measured by various standards all accessible via the mesh metrics in ANSYS Meshing software. The two most relevant control mechanisms are the aspect ratio (AR) and the skewness. The former is the ratio of the longest to the shortest edges in a cell. To ensure the best results the value should be as low as possible, with a minimum value of 1. A large AR can result in unacceptably high interpolation errors causing the simulation to be untrustworthy. The skewness is also related to the equilaterality of triangles. The ANSYS Meshing documentation provides us a list representing the mesh quality in accordance with the value of skewness, as shown in Table 10.

Table 10: Cell quality with respect to the skewness

Value of Skewness	**Cell Quality**
1	degenerate
0.9 — <1	bad (sliver)
0.75 — 0.9	poor
0.5 — 0.75	fair
0.25 — 0.5	good
>0 — 0.25	excellent
0	equilateral

As we can see, elements with a value of 1 should be avoided at all cost. Simulations would not start if this kind of elements were present. Too much highly skewed elements are not wanted. This would lead again to invalid results.

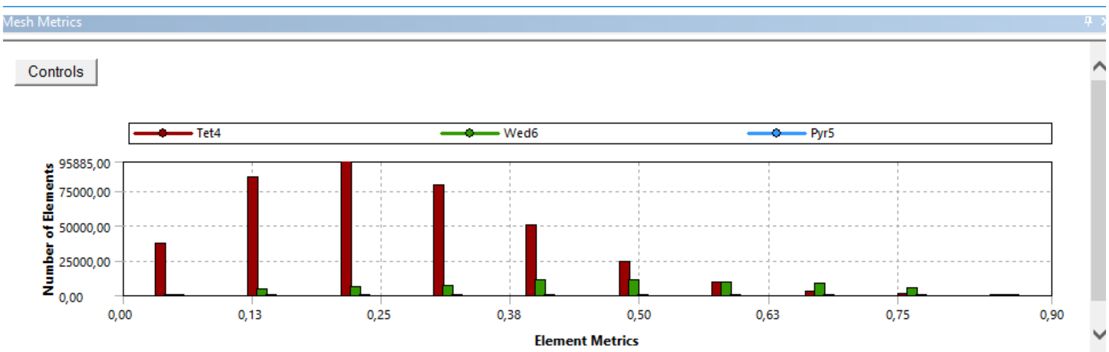


Figure 12: Mesh metrics - Skewness

The clear majority of elements have a quality that varies from good to excellent, while the worst parts of the mesh have a skewness of 0.89 to 0.90 (exceptions are cases 8, 14 and 21, the worst elements have a skewness of respectively 0.93; 0.93 and 0.97),, in contrast to the 0.98 of the original.

5.3 Model setup

The model setup is defined in the CFX – Pre-software program. This section will provide the reader inside in the boundary conditions, the turbulence model, fluid models, initial conditions etc.
In the outline of the CFX-Pre, two domains are defined. One is the rotating part, and the other the static (both indicated on Fig. 13). As for the Basic Settings in the turning part, the centre of rotation is the middle of the water wheel. The fluids and particles is defined as air (at 25°C) and water, under a pressure of 1 atm. We choose for a ‘Buoyant Model’ where the gravity is chosen at -9.81 m/s², according to the correct Axis, and the ‘Buoyancy Reference Density’ equals 1.2 kg/m³. The ‘Angular Velocity’ is entered as an expression under the name ‘rotatingspeed’. This enables us to define this value as an input parameter.
The selected ‘Turbulence Model’ under Fluid Models is ‘K-epsilon’, where the model is defined as homogeneous, since the simulation consists of 2 phases, so the fluids water and air are defined. The fluid temperature is set on 25°C.
As for the initial conditions, we start from a stationary situation where all velocity values are set at 0 m/s, as well as the relative pressure (0 Pa). The ‘Turbulence’ is set on Medium, which represents an intensity of 5%. Initially, the environment of the rotating part consists of 100% air and no water.
Inside this domain, the blades, the plates and edges are assigned the boundary condition type ‘Wall’ (no slip and smooth), while the boundary type ‘symmetry’ is assigned to the face of the rotating part situated on the axis of symmetry.

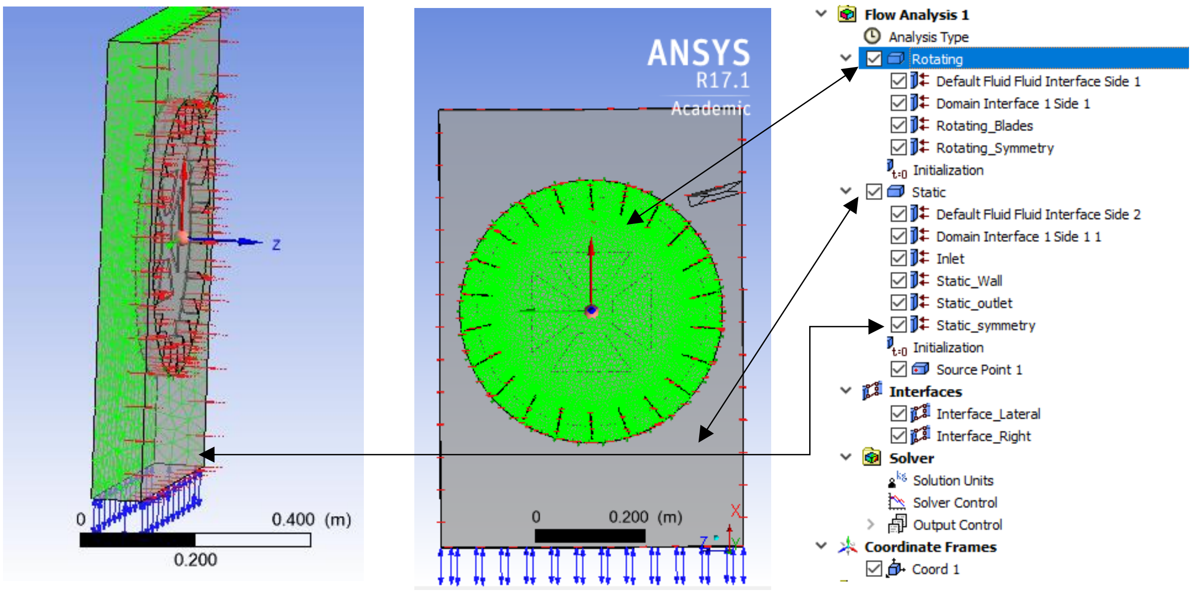


Figure 13: Left:’ Static_Wall’ highlighted in green; middle: boundary condition type ‘Symmetry. This is true for the rotating as well as the static part; right: tree outline of the different boundary conditions.

The coordinate system defined under ‘Coordinate Frames’ (see Figure 13) is very important. This makes sure that the torque curves remain oscillation free. An oscillated curve is shown in Figure 14. The orientation of the axes was later changed to get positive results regarding the relevant torque, represented by the orange curve.

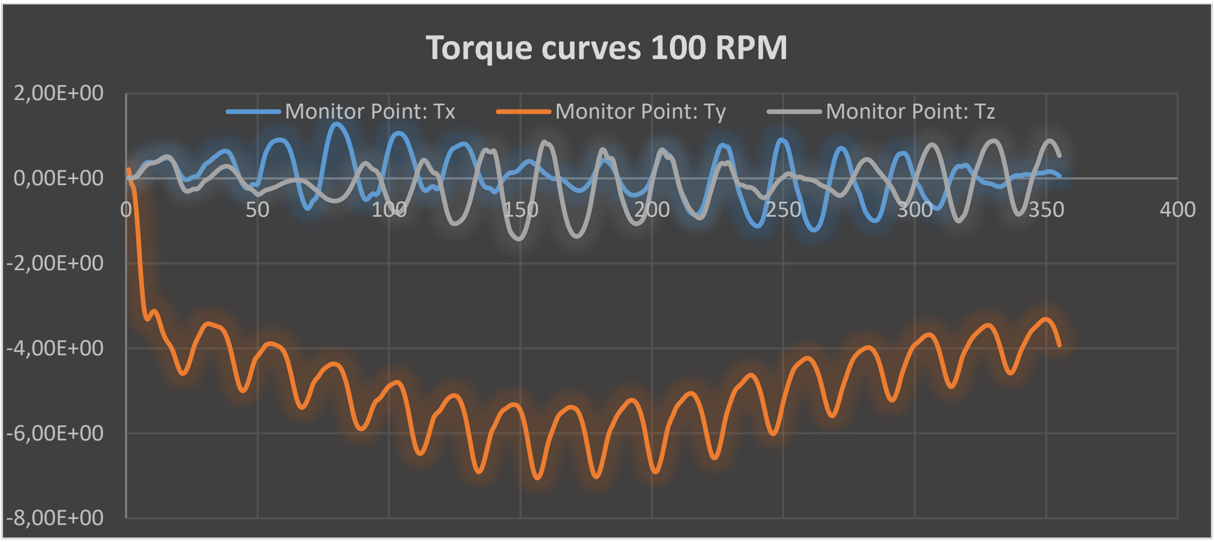


Figure 14: Oscillating torque curve

Inside the static domain, the remaining parts are defined. The inlet face is defined as an inlet boundary, where the speed, of the water is a constant value for our simulations, equals 14.39m/s, - as calculated in formula (7) - for a diameter of 20.7 mm. The volume fraction consists of 100% water.

The 'Wall' type of boundary is reserved for the edges of the box surrounding the turbine. This means that the 'Static_Wall'(this name envelops the Static_top, Static_Right, Static_Back, Static_Front and the Static_Tube_Lateral) – see Figure 12 – has this condition assigned, with 'No Slip Wall' and 'Smooth Wall' as details. The static outlet represents the bottom of the box, and is open so that water can flow away. Thus, it has the boundary condition 'Opening' assigned, it has 100% air in the beginning and a relative pressure of 0 Pa, so equal to the atmosphere. The face of symmetry gets the corresponding boundary type, as was the case with the rotational part. The only difference in the Basic Settings of the Static Domain is that we select 'Stationary' by the Domain Motion option. The initial conditions are the same, as well as all other values and option boxes.

The analysis type is 'Transient', since the power output will never converge to a constant value, since it depends on the position of the blades and it is thus a function of time. To carry out proper simulations, we will make use of a constant degree change for the different timesteps, as already mentioned in the previous chapter. Every degree will represent one time step, which allows us to calculate the time step value for different peripheral velocities:

$$\Delta\theta = 1°$$

$$\Delta\theta = \omega\ \Delta t$$

$$where\ \omega\ is\ the\ peripheral\ speed\ and\ \Delta t\ the\ time\ step.$$

$$\Delta t = \frac{\Delta\theta}{\omega} = \frac{1°}{\omega}$$

$$\omega = n\ \frac{2\pi}{60}\quad \left[\frac{rad}{s}\right]$$

$$\Rightarrow\ \Delta t = \frac{\Delta\theta}{\Delta\omega} = \frac{\Delta\theta * \frac{2\pi}{360}}{n * \frac{2\pi}{60}} = \frac{\Delta\theta}{n} * \frac{60}{360}$$

$$\Rightarrow\ \Delta t = \frac{1}{6 * n}$$

For every time step 10 iterations will be carried out to ensure an accurate and an error insensitive result, which is done in the solver control. The Transient Scheme that is utilised is the Second

Order Backward Euler, since this gives us the highest accuracy. This is also the reason why we selected 'High resolution' under 'Turbulence Numerics'.
Small adjustments are made to the original setup of the simulations. Firstly, we define a new coordinate system to make sure the origin is situated in the middle of the rotating part, as shown in Figure 15. If we do not do that we get oscillated, and thus invalid, results represented by the orange curve in Figure 14. The direction of the axis is also modified to gain positive torque values in a later stage.

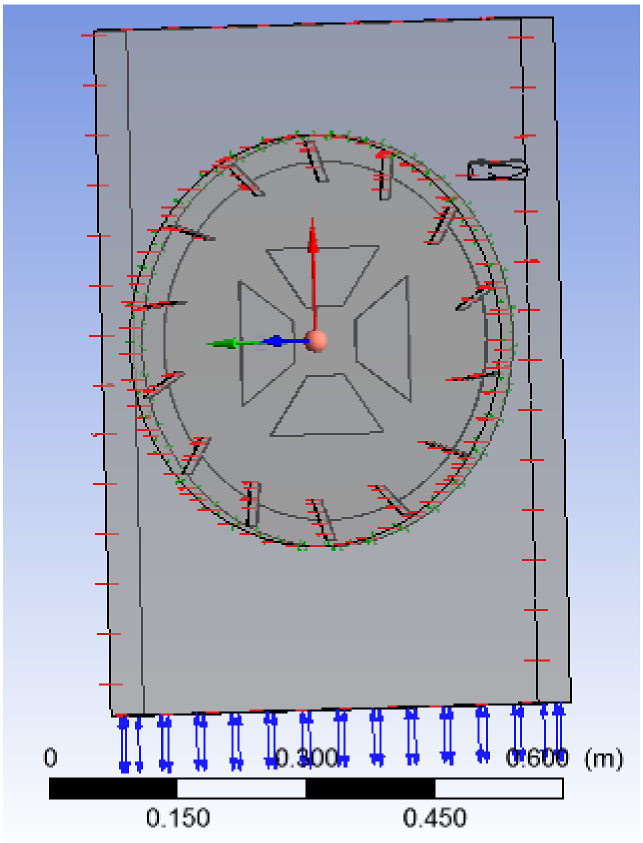


Figure 15: Setup for run 1. This setup can be found in chapter 4.3, Table 9.

The second adjustment is that we will save the situation of the process every 20 timesteps to save up on data space. This is done under the 'Output Control', at the tab 'Transient Results'. We will also manage the expressions for the Torque there, under the tab 'Monitor', where the torques are assigned to the new coordinate system.

6 Results and discussion

After preparing the different setups from Table 9 in the parameter set in ANSYS Workbench, we can process all the different cases. Figure 16 shows the input values for case 5 of the simulations (see Table 9).

Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	200 (B1)			
4	P7	rotatingspeed	-200	rev min^-1
5	P8	timestepvalu	0,00041667	s
6	P9	totaltime	0,6	s
7	Geometry (A1)			
8	P14	Vertical_tube_position	191,95	mm
9	P15	Horizontal_tube_distance	100	mm
10	P16	Angle_Inlet	-20	degree
11	P19	Number_Of_Blades	21	
12	P21	Inclination_angle_blades	180	degree
*	New input parameter	New name	New expression	
14	Output Parameters			
15	200 (B1)			
16	P10	TorqueZ	0,80269	J
17	P11	TorqueY	-4,9351	J
18	P12	TorqueX	0,11675	J

Figure 15: Outline of the different parameters. Note that the 'rotatingspeed' is set as an input parameter.

6.1 Results of Taguchi method

The results of all the cases obtained from the finite volume simulations are shown in Table 11 on page 33. The power output can be also found. The negative values mean that we have a negative torque, which indicates that the wheel turns faster than the water jet speed projected in the angular coordinate. This should not pose a problem since we will normalise the results using the formula for the S/N ratio, which is a quadratic function.

The lower power values in Table 11 can be explained by looking at the configuration. The combination of an inlet with a sharp angle, with blades that are also heavily inclined (as in Fig. 17 and 18) can make that the water reflects immediately off the blades, rather than pushing them and thereby conveying energy. On the right side, we included the best run of the Taguchi method, to see the contrast. It is clear that, for an optimal power output, the impact should almost be perpendicular, and that the inclination of the blades, as well as that of the inlet tube, has a big impact.

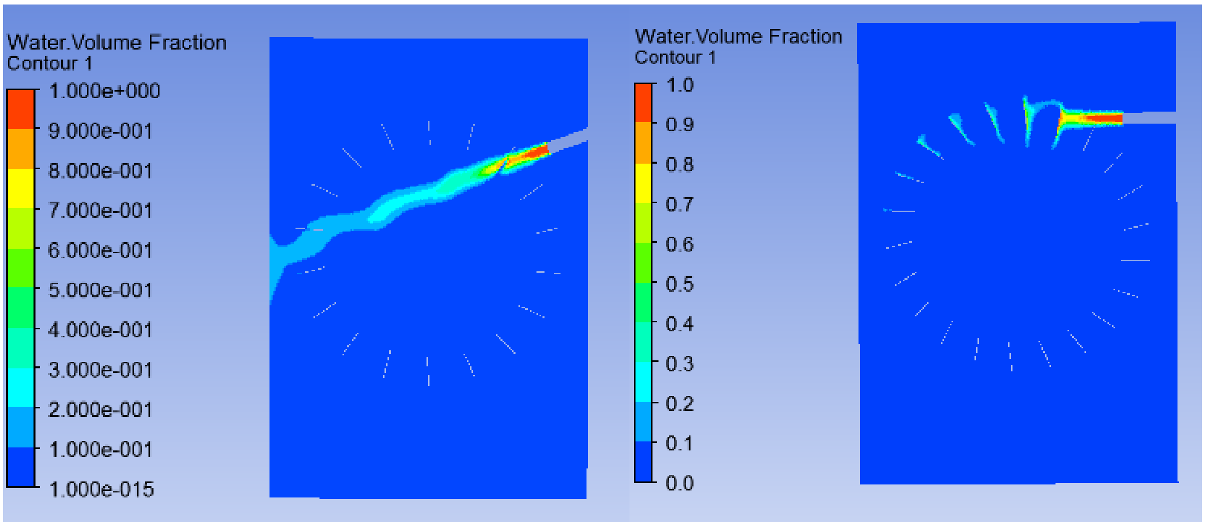


Figure 17: Left: the result of run 22. We can see little impact on the blades and a lot of water loss. The straight line of the water implies that the turning velocity of 350 rpm is too high for the rest of the configuration; Right: result of run 12. A better configuration makes sure the waterjet has a near perpendicular impact and generates more torque.

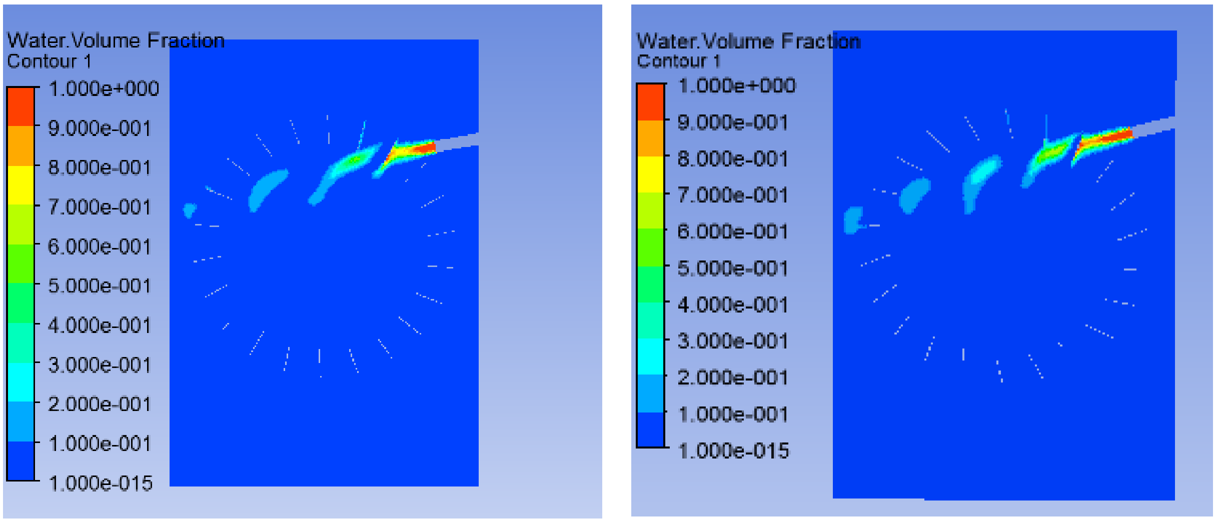


Figure 18: Left: Result of run 6, snap taken at timestep 80. The water clearly reflects of the blades and the turbine suffers inevitably a loss of power output. The configuration also allows too much water to be spilled, represented by the water fraction centrally that aligns with the inlet tube inclination. Right: Result of run 18, snap taken at timestep 80. The less favourable inclination of the blades make sure even more water is lost and this reduces in a lower power output.

The following figures provide snaps of the pressure areas on the blades to gain a better understanding of the impact of the configuration on the power output. The pictures represent the evolution from timestep 40 until timestep 100. We include the same runs as above, starting with Figure 19 that will represent run 6.

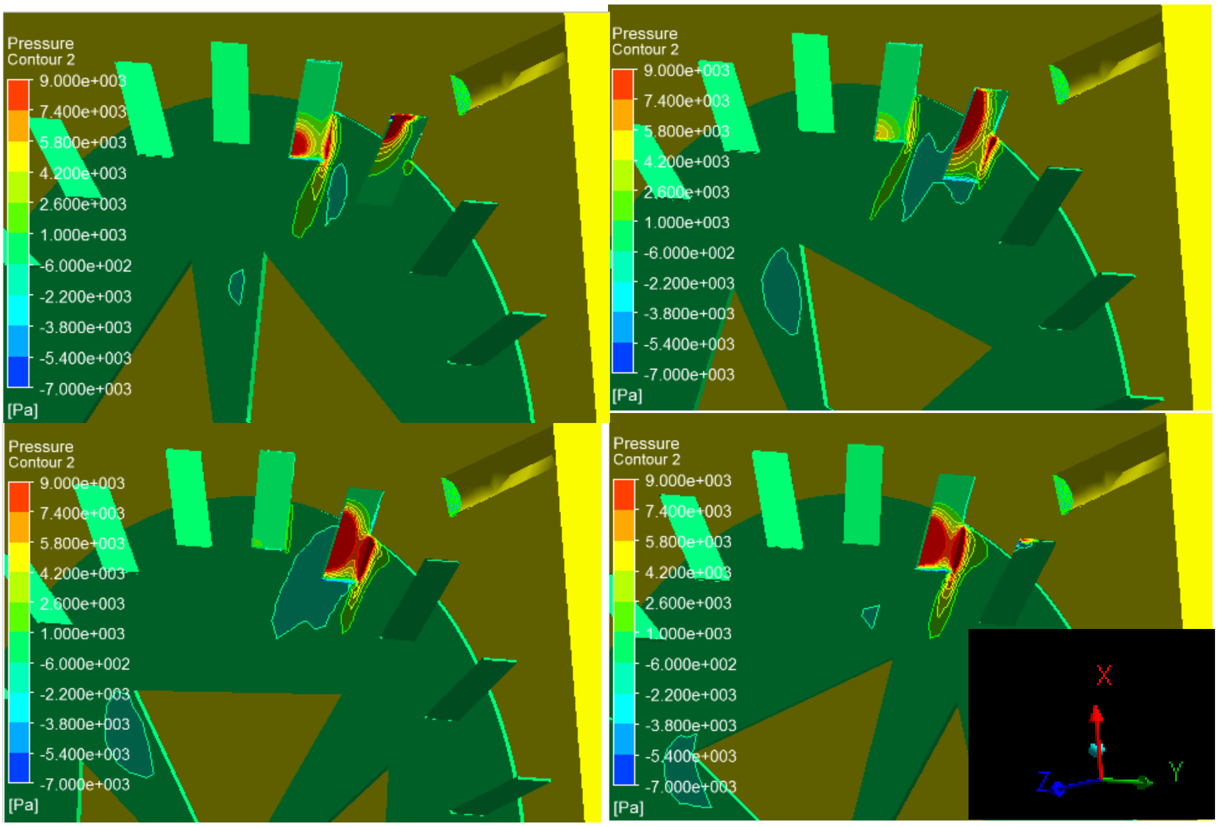
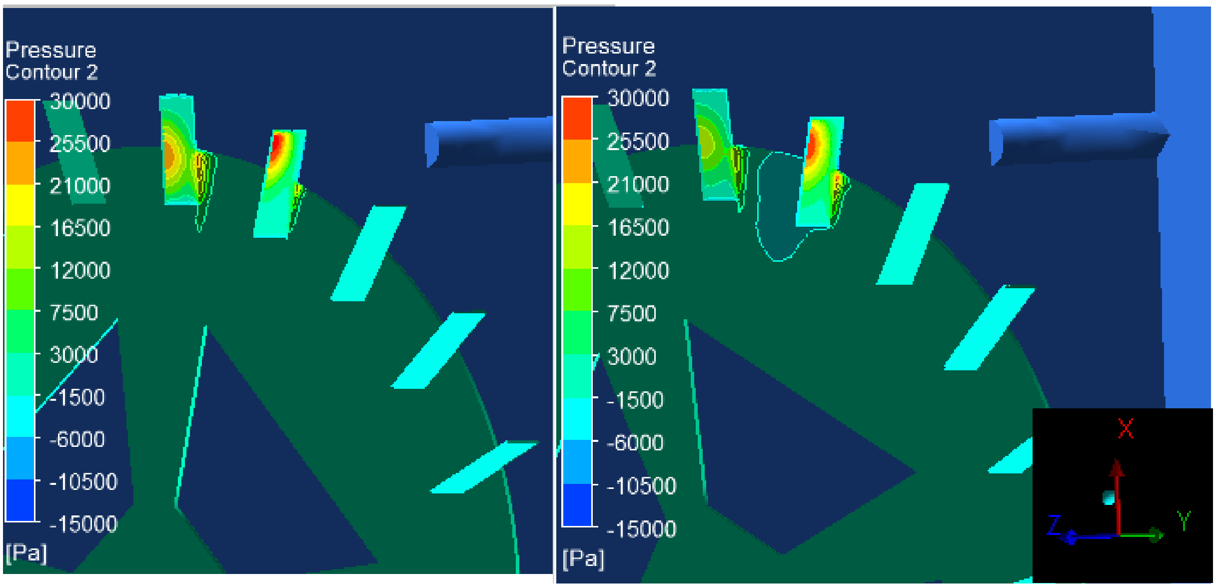


Figure 19: Pressure levels for run 6 of the Taguchi method. Upper left shows the situation at time step 40. Upper right 60, bottom left time step 80 and bottom right represents time step 100. Pressure levels have a maximum of 9000 Pa.

Next is the best run from the Taguchi setup, as shown in Figure 20. We notice pressure values that are notably higher, as well as almost no pressure coming from water losses in between the blades;



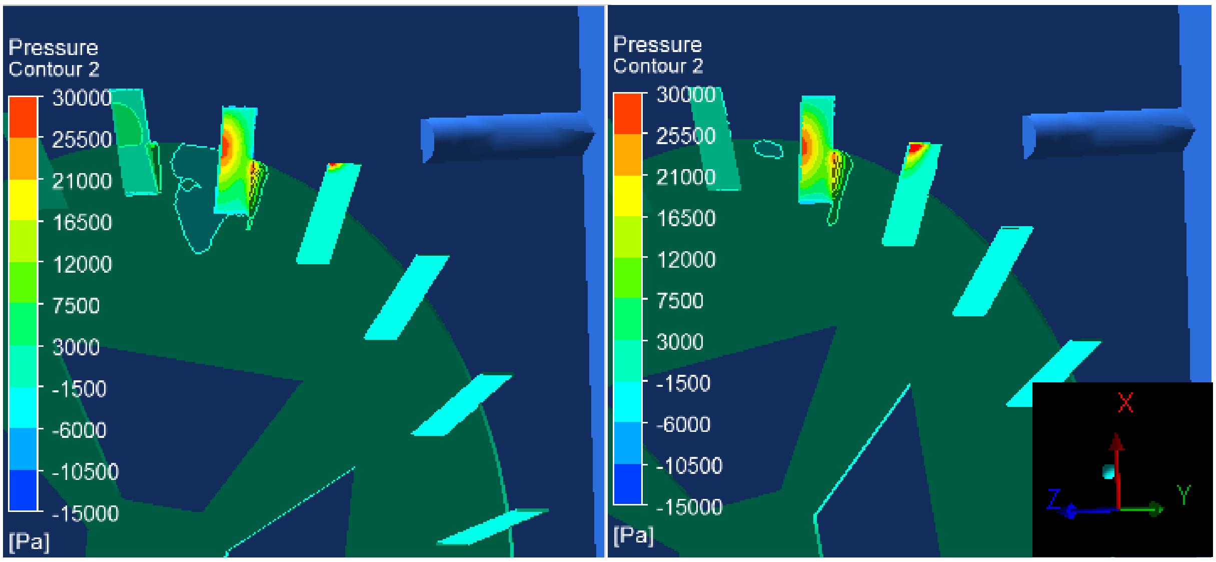


Figure 20: Pressure contour of run 12. Less water loss on the rotating part, which - along with other differences - results in a higher power output. The water losses that are present situate themselves around the blades.

Run 18 is shown on Figure 21. The maximum pressure is a lot higher than in Figure 20, but it is also a lot more concentrated.

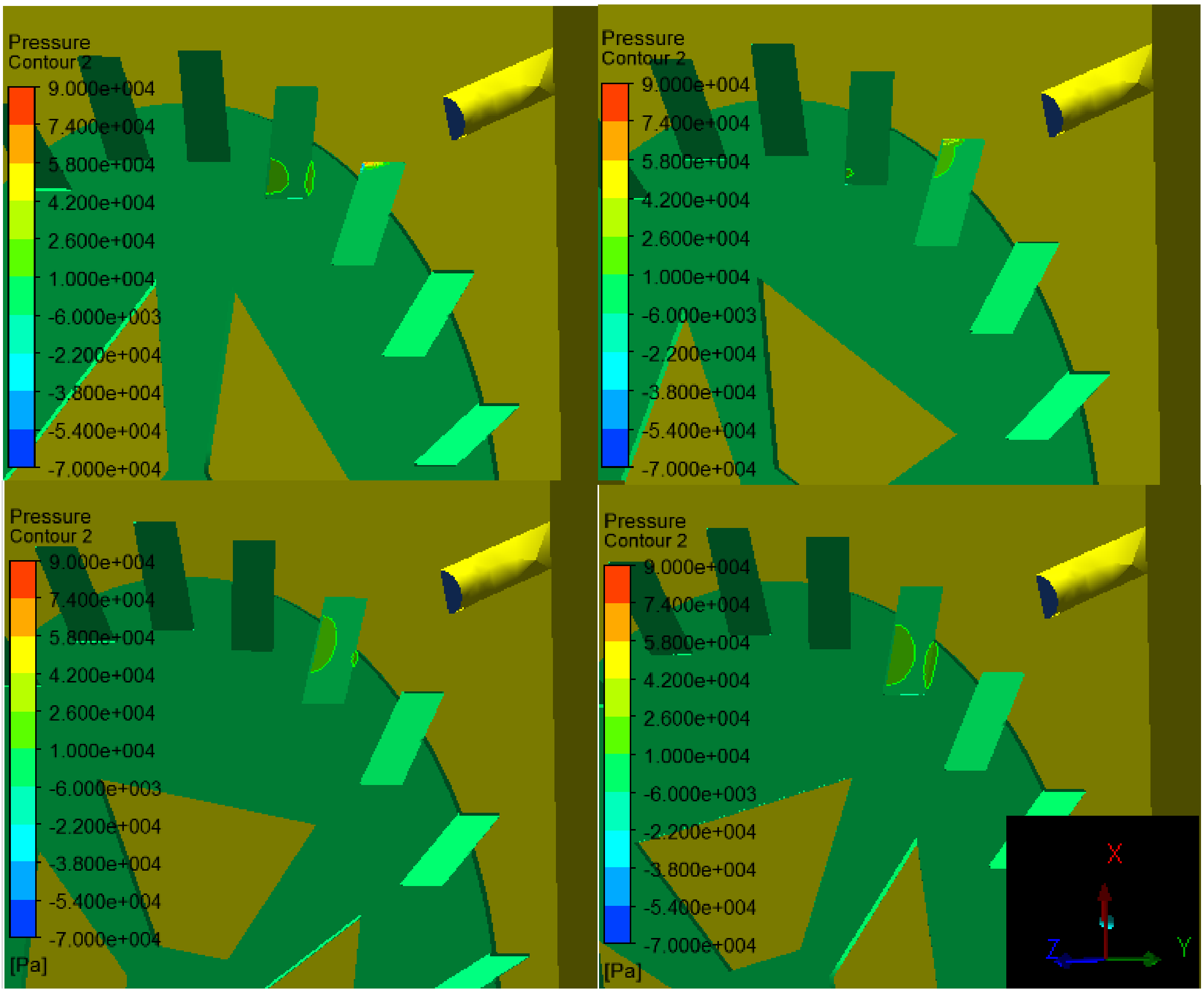


Figure 21: Pressure levels of run 18.

Figure 22 shows the pressure snaps from run 22. Clearly there is not enough impact on the blades to generate torque.

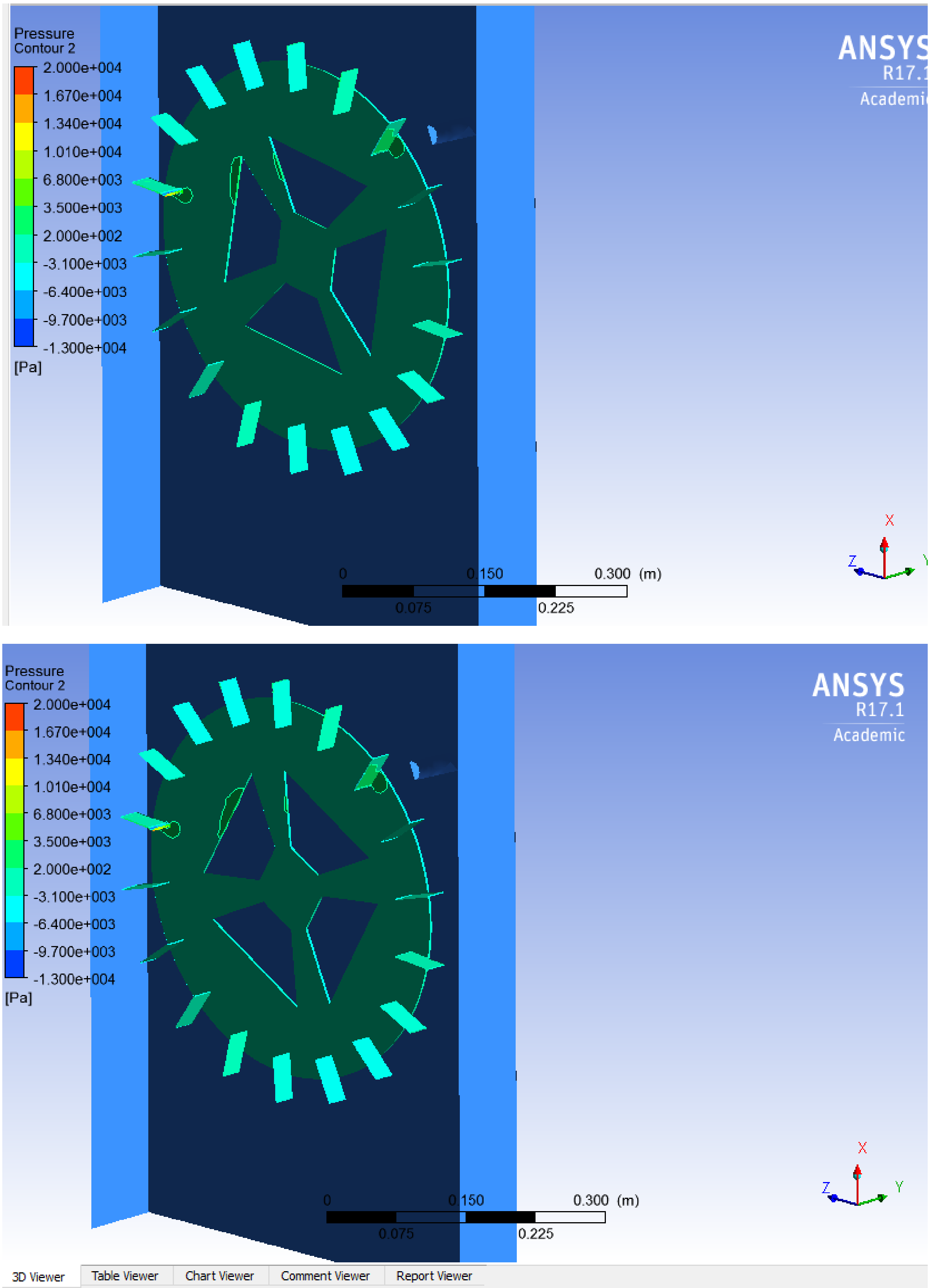


Figure 22: Run 22. The highest pressure is found on the left side of the rotating wheel. We only included two since there is no significant difference.

In Figure 23 and Figure 24 we show the normalised superficial velocity (this is the flow velocity calculated as if it was the only fluid or phase in this area or cross-section) of air of runs 6 and 12, while Figures 25 and 26 show their respective vectors of water.

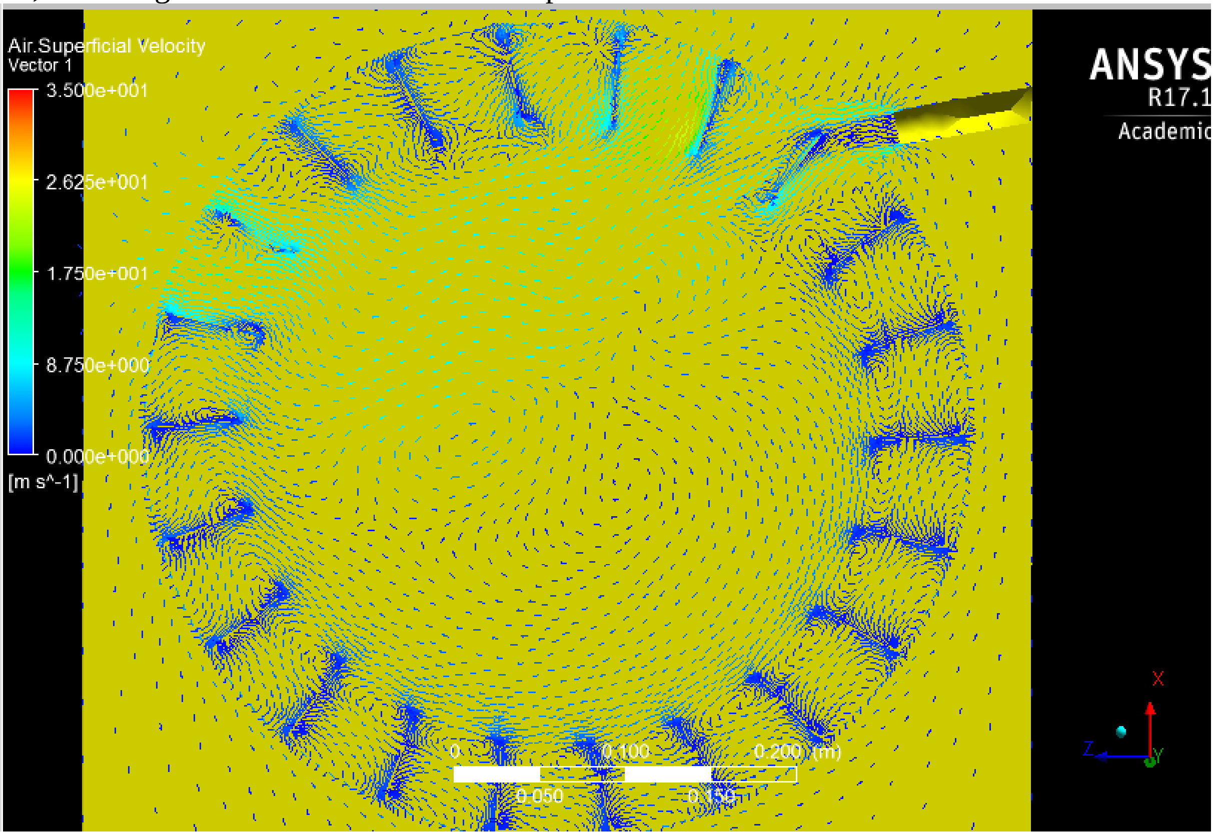


Figure 23: Air superficial velocity of run 6, snap is taken from timestep 40.

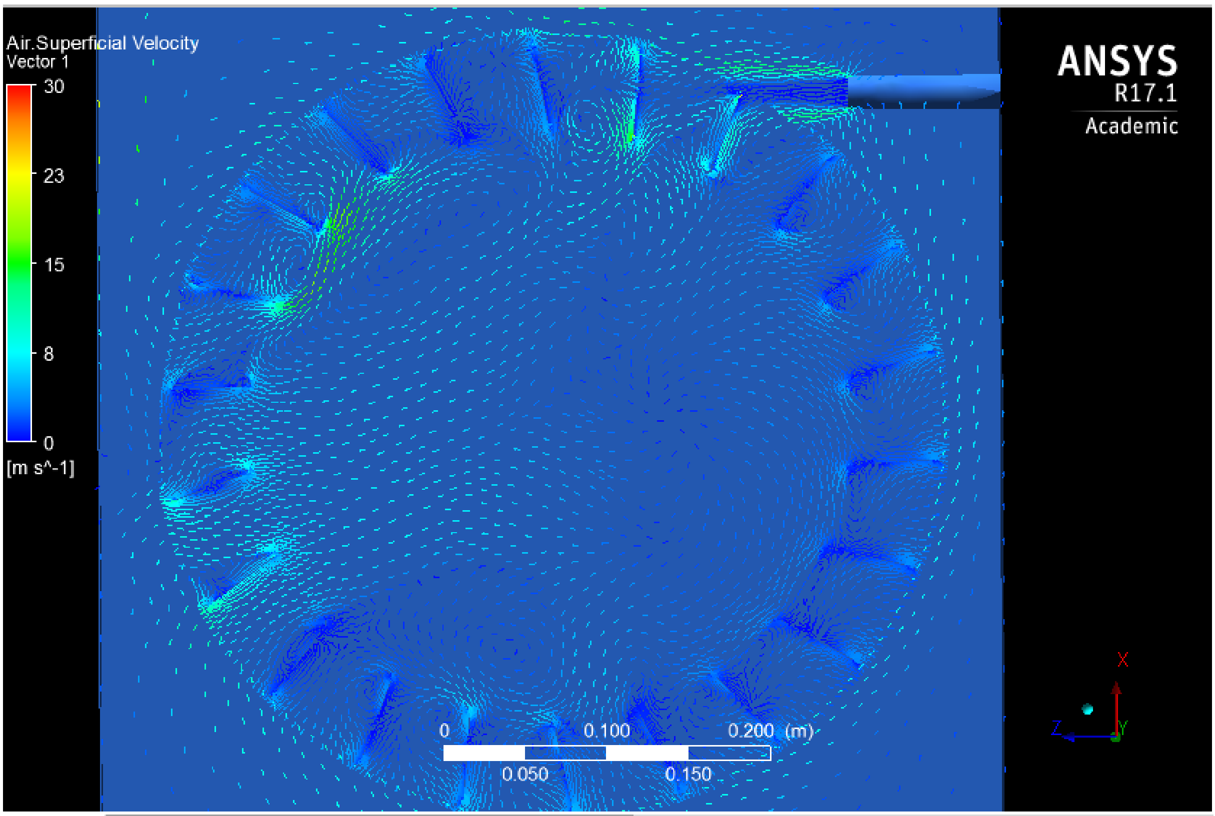


Figure 24: Air vectors of run 12, snap taken at timestep 100.

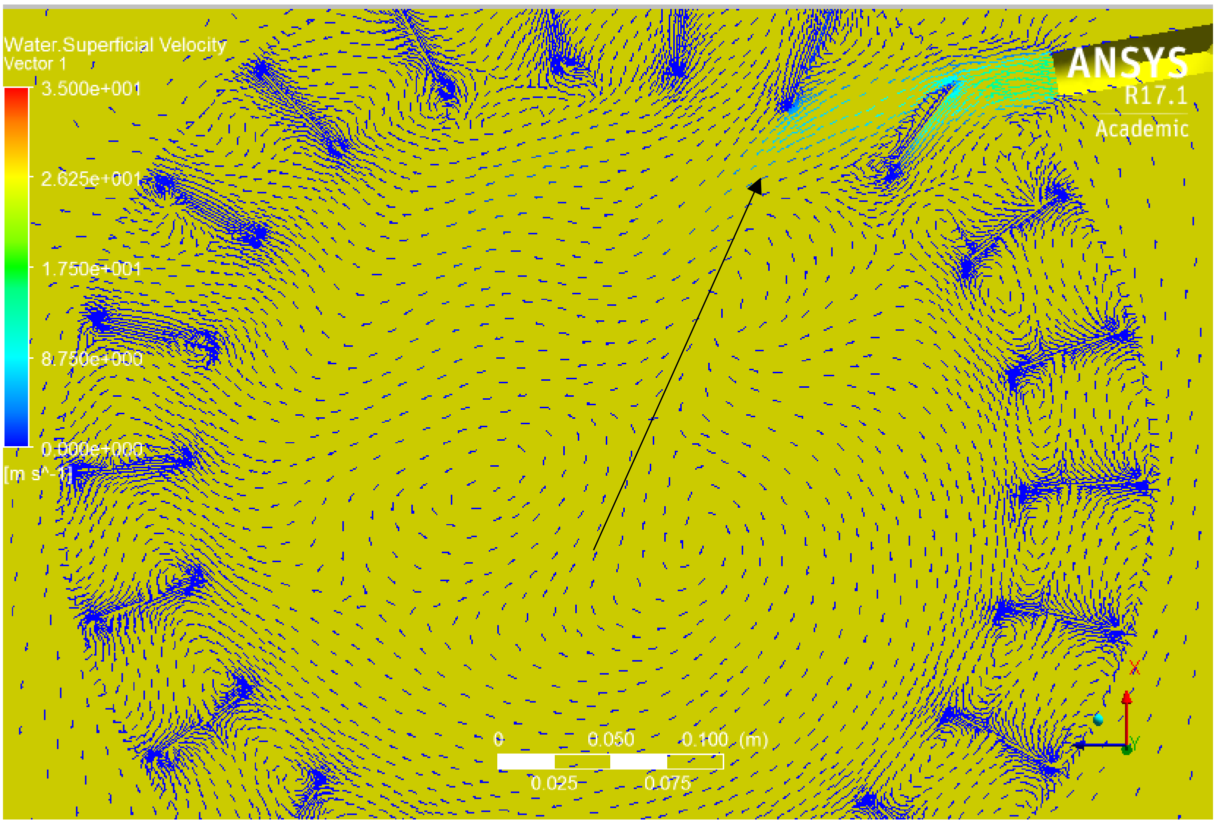


Figure 25: Water superficial velocity of run 6, snap taken at timestep 40. It shows clearly the loss of water in between the blades.

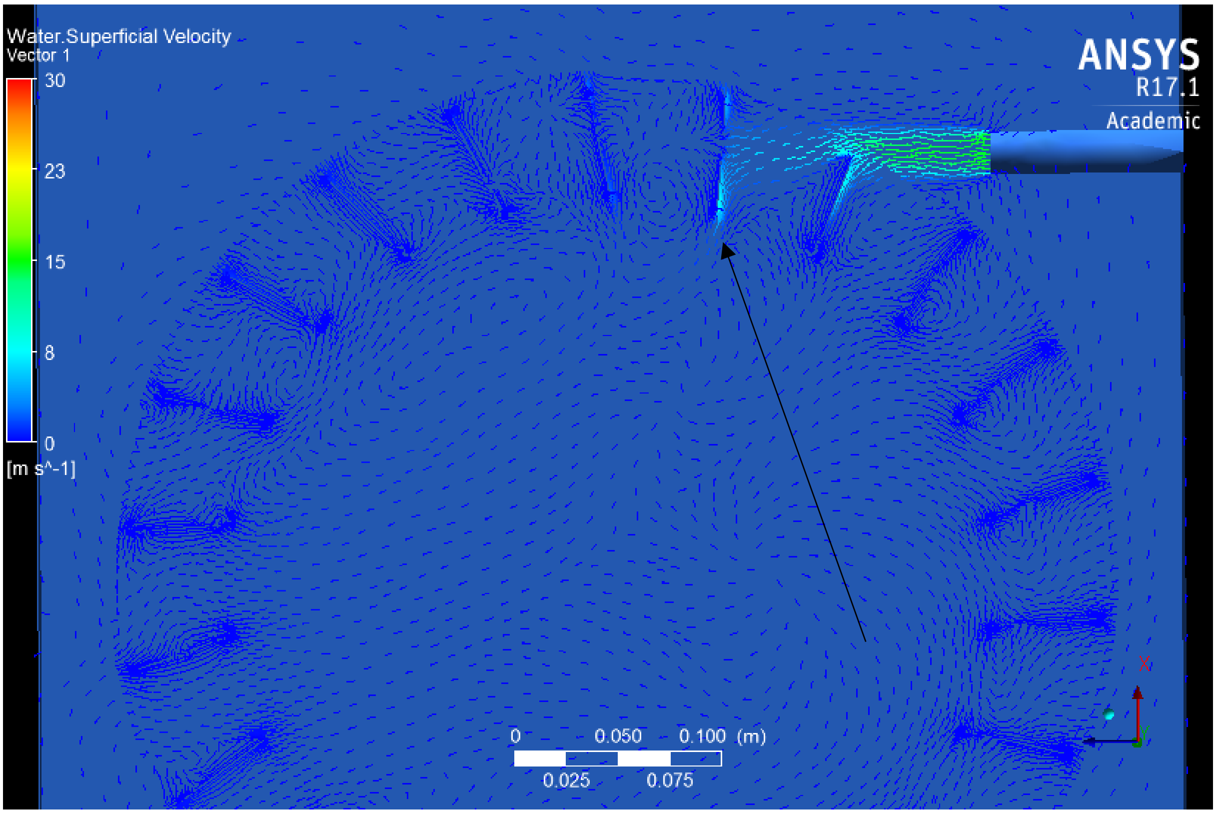


Figure 26: Water vectors of run 12, snap taken at timestep 100. Next to no loss of water, although we can see a slight spill of water at the bottom of one of the blades.

Table 11: Results (are shown in italic) of the Taguchi cases. The turbine efficiency is calculated using the value of Eq. 10.

	Factors						Results		
	A	*B*	*C*	*D*	*E*	*F*			
Run	**Velocity**	**VTP**	**HTD**	**IA**	**Blade inclination**	**Number of blades**	**Power**	**Turbine efficiency**	**S/N ratio**
	rpm	mm	mm	°	°		W	%	
1	200	146.95	73	0	160	13	*143.80*	*28.14*	*4.32*
2	200	158.2	79.75	-5	165	15	*136.20*	*26.65*	*4.27*
3	200	169.45	86.5	-10	170	17	*108.20*	*21.17*	*4.07*
4	200	180.7	93.25	-15	175	19	*79.80*	*15.62*	*3.80*
5	200	191.95	100	-20	180	21	*53.40*	*10.45*	*3.46*
6	237.5	146.95	79.75	-10	175	21	*65.20*	*12.76*	*3.63*
7	237.5	158.2	86.5	-15	180	13	*28.80*	*5.64*	*2.92*
8	237.5	169.45	93.25	-20	160	15	*14.98*	*2.93*	*2.35*
9	237.5	180.7	100	0	165	17	*105.20*	*20.59*	*4.04*
10	237.5	191.95	73	-5	170	19	*166.00*	*32.49*	*4.44*
11	275	146.95	86.5	-20	165	19	*33.80*	*6.61*	*3.06*
12	275	158.2	93.25	0	170	21	*186.80*	*36.56*	*4.54*
13	275	169.45	100	-5	175	13	*159.20*	*31.15*	*4.40*
14	275	180.7	73	-10	180	15	*114.80*	*22.47*	*4.12*
15	275	191.95	79.75	-15	160	17	*81.00*	*15.85*	*3.82*
16	312.5	146.95	93.25	-5	180	17	*83.80*	*16.40*	*3.85*
17	312.5	158.2	100	-10	160	19	*54.80*	*10.72*	*3.48*
18	312.5	169.45	73	-15	165	21	*13.44*	*2.63*	*2.26*
19	312.5	180.7	79.75	-20	170	13	*-14.04*	*-2.75*	*2.29*
20	312.5	191.95	86.5	0	175	15	*23.80*	*4.66*	*2.75*
21	350	146.95	100	-15	170	15	*-56.00*	*-10.96*	*3.50*
22	350	158.2	73	-20	175	17	*-58.40*	*-11.43*	*3.53*
23	350	169.45	79.75	0	180	19	*160.20*	*31.35*	*4.41*
24	350	180.7	86.5	-5	160	21	*135.80*	*26.58*	*4.27*
25	350	191.95	93.25	-10	165	13	*105.40*	*20.63*	*4.05*

Note 2: VTP = vertical tube position; HTD = horizontal tube distance; IA: inlet angle

We then calculated the mean S/N ratios using formula (14), with the results shown in Table 12.

The given values are calculated by taking the average of the corresponding level and factor. For example, the mean S/N ratio for the factor B level 2 in terms of power output is equal to (4.268 + 2.919 + 4.543 + 3.478 + 3.533)/5 = 3.748 (equal to the S/N values in Table 11 for runs 2, 7, 12, 17 and 22). Thus, the table was filled. To identify the factors with the biggest influence on the performance of the turbine, we search for the maximum and minimum value in each row. The higher the difference, the greater the parameter's importance regarding the power output.

Table 12: Normalisation of the results followed by max- and minimum values of each row.

	Level							
Factor	1	2	3	4	5	Max	Min	Δ
A	3.982	3.477	3.988	2.926	3.950	3.988	2.926	1.063
B	3.669	3.748	3.498	3.706	3.702	3.748	3.498	0.250
C	3.733	3.684	3.413	3.718	3.775	3.775	3.413	0.363
D	4.013	4.245	3.868	3.259	2.938	4.245	2.938	1.307
E	3.645	3.535	3.769	3.624	3.750	3.769	3.535	0.234
F	3.596	3.398	3.862	3.838	3.630	3.862	3.398	0.464

Using Table 12 we can visualise the results in a graph, as shown in Figure 27.

Figure 27: Visualised results out of Table 12. Note that the Y-axis starts at '2', this is to make the difference in height of the lines clearer.

As stated above the difference in height is important. Figure 27 makes clear that this is surely the case for factors D (the inclination of the inlet tube) and A (the rotating speed). We can also highlight the importance per factor by plotting the difference in maximum and minimum value, as done in Figure 28.

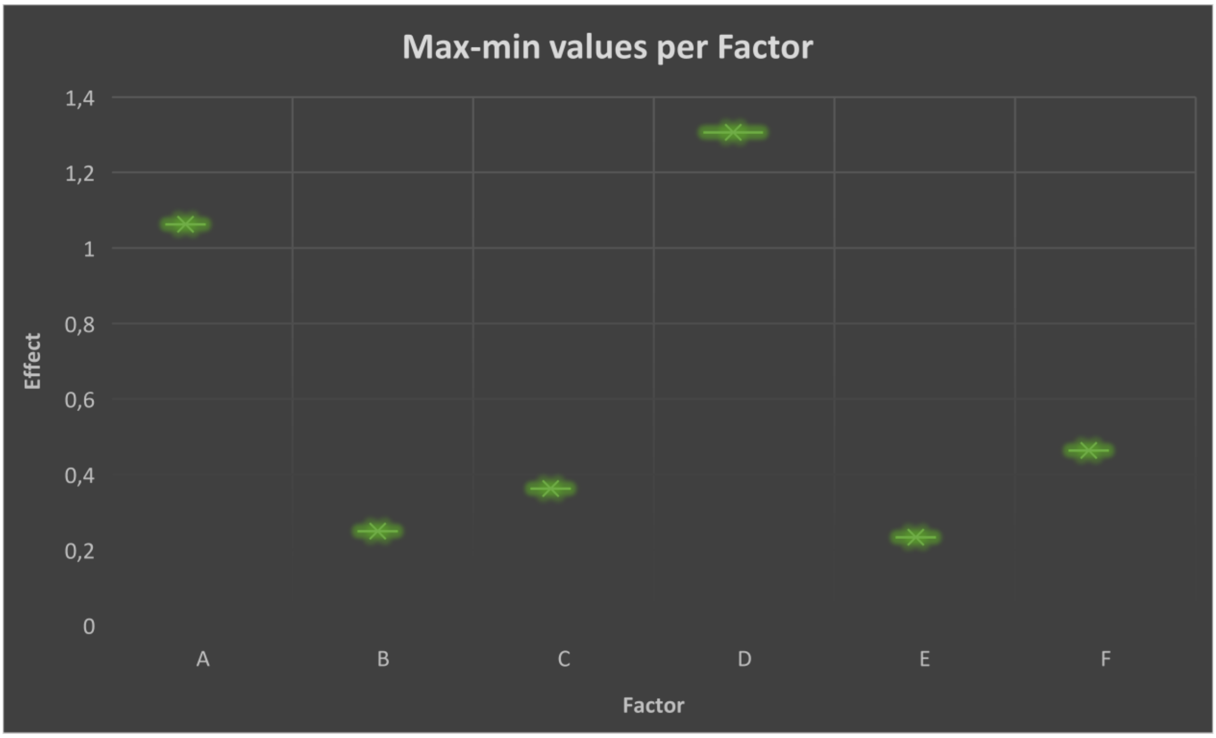


Figure 28: Maximum value per factor. These values per parameter combined are supposed to result in an optimised performance of the horizontal axis water jet turbine. The values can be found in the last column of Table 12.

The last column of Table 12 gives us the information needed for the optimisation. The most important features according to the used method are in order: Inclination inlet tube (factor D) > Rotating speed (factor A) > Number of blades (factor F) > Horizontal tube distance (factor C) > Vertical tube position (factor B) > Blade inclination (factor E).

The next step is then to do a simulation with the maximum value for the combination Factor-Level. Table 13 highlights the values that are supposed to give a close to ideal setup regarding power output:

Table 13: The highlighted values represent a proposal of an optimised configuration.

Factor	Parameter	Units	level				
			1	2	3	4	5
A	Rotating speed	rpm	200	237.5	275	312.5	350
B	Vertical tube position	mm	146.95	158.2	169.45	180.7	191.95
C	Horizontal tube distance	mm	73	79.75	86.5	93.25	100
D	Angle inlet	Degrees	0	-5	-10	-15	-20
E	Blade inclination	Degrees	160	165	170	175	180
F	Number of blades	/	14	16	18	20	22

We then did five simulations with this setup where we varied the rotating speed from 270 rpm to 280 rpm with step size 2 rpm. Figure 29 shows the resulting torque curves of the first (and best) run of this interval. The result with the highest power output is shown in Table 14.

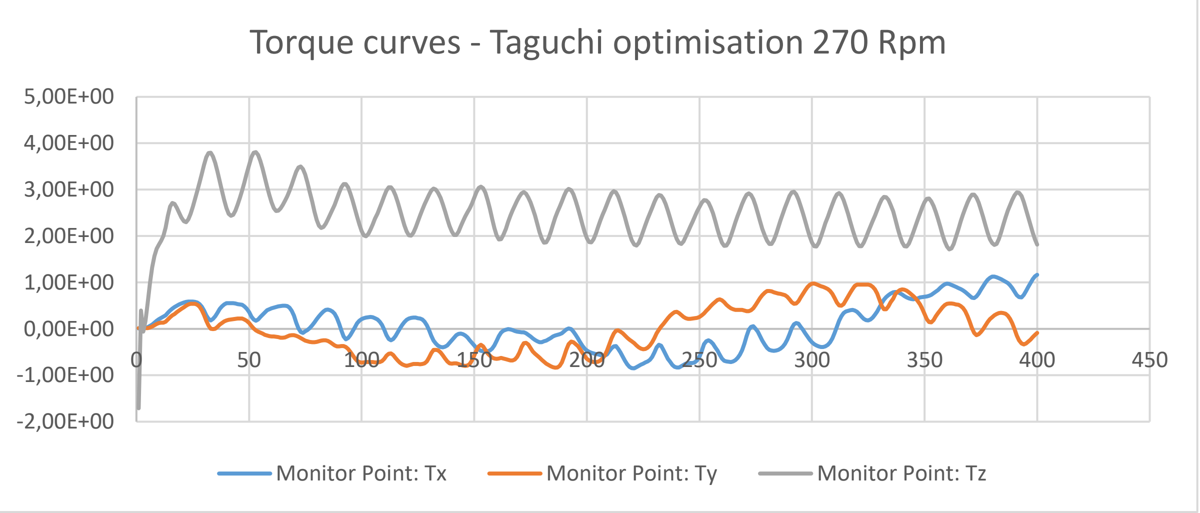


Figure 29: Torque curves of the 'optimised' run. Values should be doubled given the symmetry.

Table 14: Power output with ideal setup according to the method.

Velocity=	270	rpm
Omega=	28.27	Rad/s
Torque=	2.51	Nm
Power=	142.07	W

Given that this power output is not close to the greatest output we can find in Table 11 (run 12; 186W), we can assume that we violated one of the rules of the Taguchi method. It is probably so that the factors are not wholly independent, more specifically the horizontal tube length shows a dependency on the height of the nozzle in combination with a strongly inclined inlet tube. We hoped to counter this by making sure we chose the ranges so that there was no interference possible between the inlet tube and the rotating part, as shown in Figure 30, but it appeared not enough.

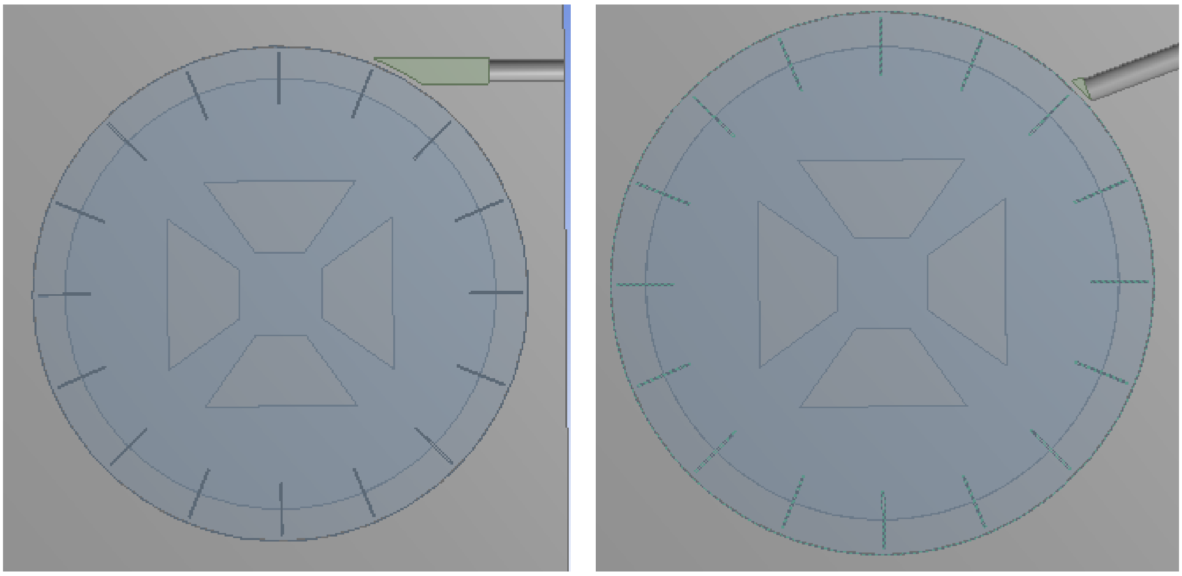


Figure 30: Left: configuration with perpendicular inlet tube with respect to the frame of the turbine; Right: sharp angle of 70° between the inlet tube and the frame, resulting in an intersection of the body of influence with the rotating part. This was never part of our any configuration. It serves as an example to show that the parameters are not completely independent from each other.

6.2 Point of best efficiency

Given the previously found results, we decided to have a look at the most eye-catching runs via the Taguchi method and applied a trial and error approach to find better results. In table 11, run 12 has the highest power output, so it is a good starting point. We also assumed that the range [260 – 280] rpm had to give the highest power output. This is based on the original power curve (see Figure 3), as well as the outcome of Taguchi.
Furthermore, we thought a blade number of 22 had to give higher results than 18. We assumed that with four more blades, less water will be wasted, so more energy will be translated to the water turbine.
These assumptions are backed up by additional tests where we kept the 'ideal' Taguchi configuration but changed the velocity. Afterwards we did some tests to see whether the blades made a big difference. The results can be found in Table 15 and are visualised in Fig. 31.

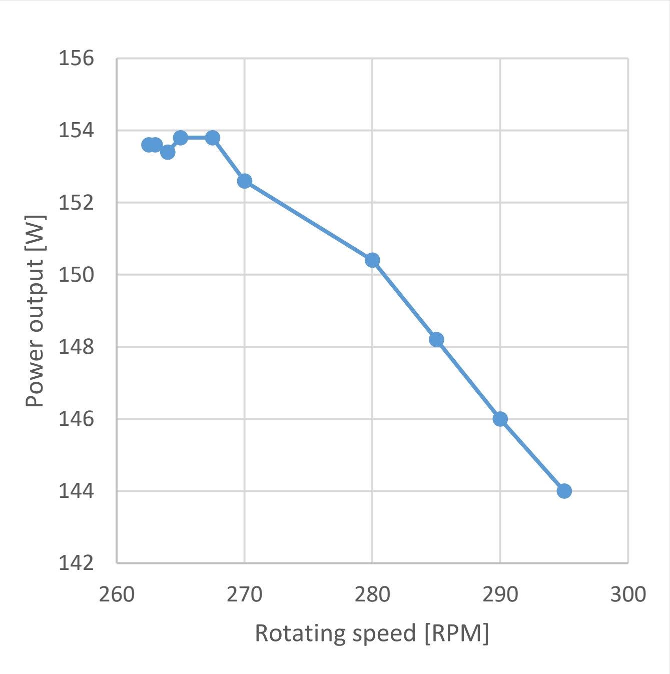


Figure 31: Visualisation of the experiments to find the ideal velocity.

Table 15: Experiments to find the best rotating speed.

Rotating speed	Power output (22b)	Efficiency
rpm	W	%
262.5	*153.6*	*30.06*
263	*153.6*	*30.06*
264	*153.4*	*30.02*
265	*153.8*	*30.10*
267.5	*153.8*	*30.10*
270	*152.6*	*29.86*
280	*150.4*	*29.43*
285	*148.2*	*29.00*
290	*146*	*28.57*
295	*144*	*28.18*

If we compare the increase of power output due to the changes in blades for the case of 270 rpm, we have an improvement of roughly 10W, which is 7 %. When running cases of 280 and 290 rpm, but with 20 blades, the cases with 22 blades still come out on top with a difference of respectively 3.4% and 2.4%.

We thus decided to continue with a rotation speed of 270 rpm and 22 blades. This still leaves 4 different parameters that can give way to a higher power output. To eliminate a lot of possibilities, we assumed that the (close to) ideal value for the inclination of the inlet tube is indeed -5°, as well as the inclination for the blades, which we set at 170. Two intermediate experiments in Table 16 with a turning velocity of 265 rpm show us a small effect of the horizontal position of the blades:

Table 16: Effect of a change in HTD, as well as the small influence of the velocity when we are simulating around the 270-rpm mark.

Velocity	VTP	HTD	Angle inlet	Blade inclination	Number of blades	Power	Efficiency
rpm	mm	mm	°	°		W	%
265	182,25	86,5	-5	170	22	*187.0*	*36.60*
265	182.25	93.25	-5	170	22	*187.8*	*36.75*
270	182.25	93.25	-5	170	22	*188*	*36.79*

The third simulation shown in Table 16, represents the highest result we could find, even after varying some parameters as shown in Table 17 on the next page:

Table 17: Results of the trial and error approach.

Run	Velocity (rpm)	VTP (mm)	HTD (mm)	Angle inlet (°)	Blade inclination (°)	Nb. of blades	Power (W)	Efficiency (%)
1	270	183.25	93.25	-5	170	22	*188*	*36.79*
2	260	183.25	93.25	-5	170	22	*186.2*	*36.44*
3	280	183.25	93.25	-5	170	22	*186*	*36.40*
4	270	183.25	93.25	0	170	22	*101.6*	*19.88*
5	270	183.25	93.25	-7.5	170	22	*179.4*	*35.11*
6	270	172	93.25	-5	170	22	*182*	*35.62*
7	270	194.5	93.25	-5	170	22	*164.6*	*32.21*
8	280	172	93.25	-5	170	22	*180.2*	*35.26*
9	280	194.5	93.25	-5	170	22	*165.2*	*32.33*
10	260	172	93.25	-5	170	22	*183.2*	*35.85*
11	260	194.5	93.25	-5	170	22	*162.8*	*31.86*
12	275	183.25	93.25	-5	170	22	*185.6*	*36.32*

This implies that the best result, regarding power output, is the first in Table 17. The highest efficiency we reached is therefore 36.79%.

To make some comparisons, we include a few graphs. One of them will be the power in function of the velocity, where we keep the AI and the VTP at respectively -5° and 172 mm. This is shown in Fig. 32.

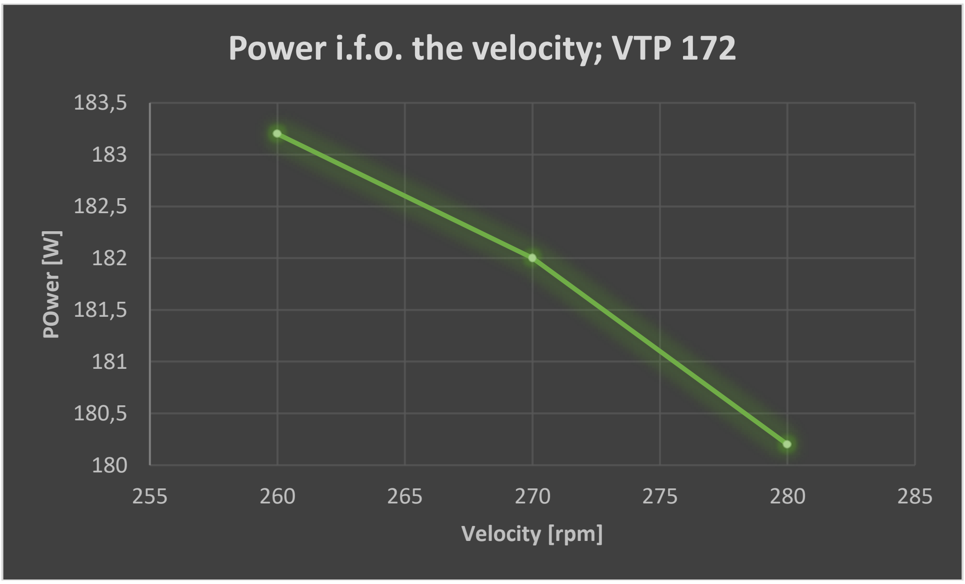


Figure 32: Power curve with the VTP set at 172mm, and the IA at -5°, number of blades: 22 and the HTD 93.25mm

The same configuration, but with the vertical tube position at 183.25 mm, gives the following Fig. 33.

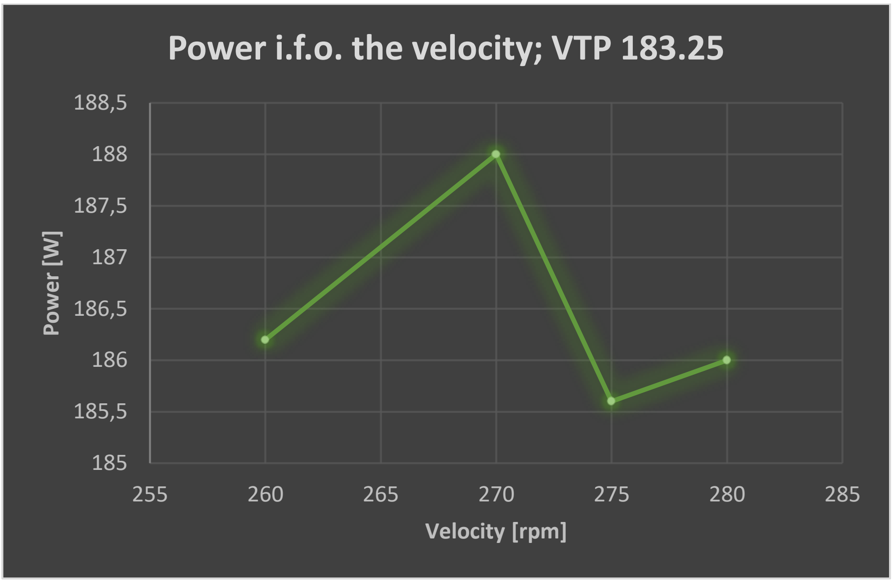


Figure 33: Power curve with the VTP at 183.25mm

The third has a VTP of 194.5, as shown in Fig. 34.

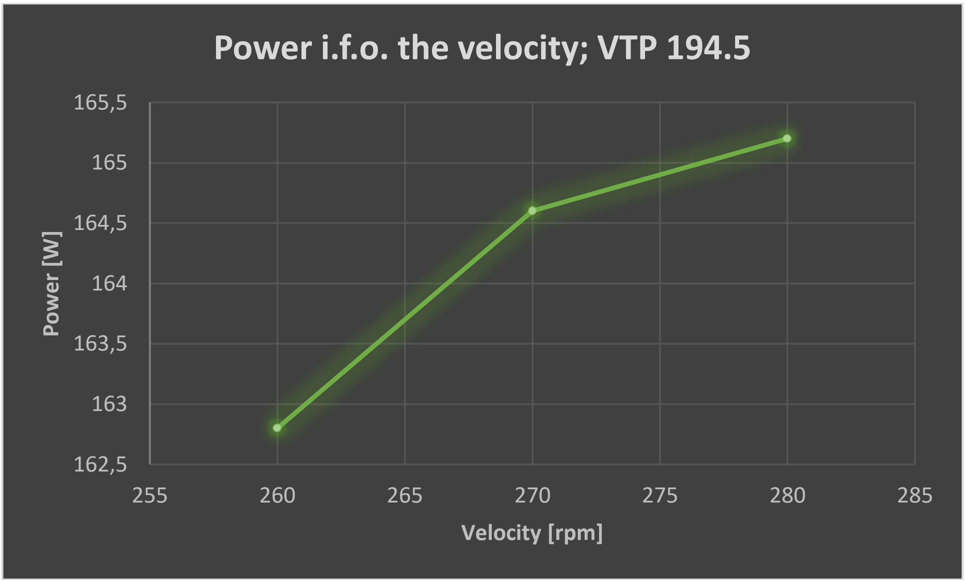


Figure 34: Power curve; VTP=194.5mm

A relative vertical tube position of 194.5 mm (= 659mm absolute height) is clearly too much in the setup of an inlet tube with an angle of -5 and de blades inclined by 170 degrees. On the other hand, the VTP of 172mm gives also lower values than the case with 183.25mm as height.

The visualisation of the influence by the angle of the inlet tube is shown in Fig. 35:

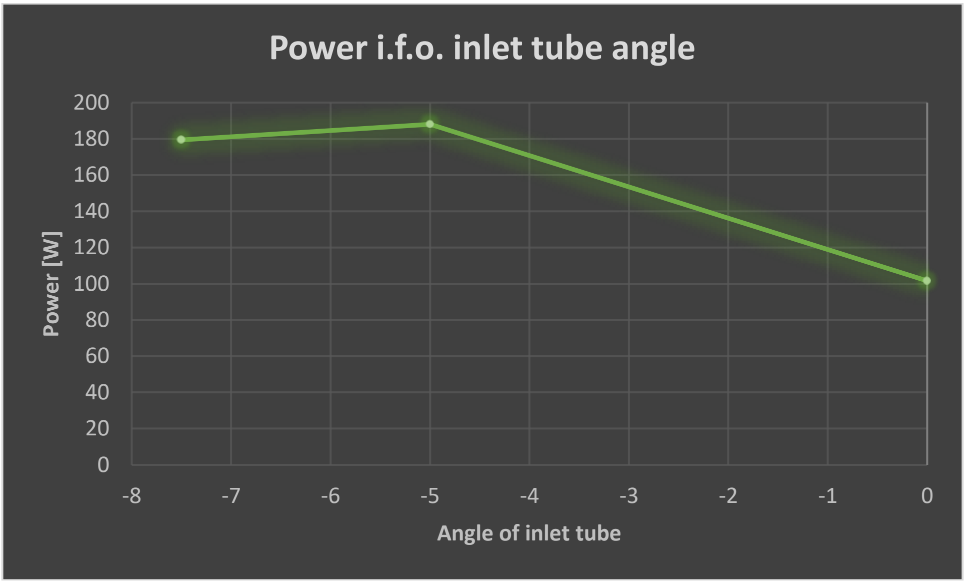


Figure 35: Influence of the angle of the inlet tube on the power curve

The inclination angle of the inlet tube clearly has a big impact in the configuration where the blades have an inclination of 170 degrees. It can be argued that from the moment the blades receive an inclination, the inlet tube angle must be adjusted accordingly, otherwise the power output value plummets.

6.3 Comparison with earlier research

The obtained values of power output are about 2% lower than the one reported from last year study [4], despite using the same geometry. There are various reasons why there might be differences, and we will try to describe them in the following paragraphs.

A first difference is the used software. We used a newer version of ANSYS, namely the R17.1, whilst the previous researcher was working with the 15.1 version. This might give small alterations in results, but only the developers know.

Another definite difference is the used mesh. We had to increase the growth rate (from 1.2 to 1.3) in order to accommodate the mesh to the limited number of elements of the student version in a more complex design (i.e., with more blades). This causes the solver to be less precise in the most important areas of the water turbine (blades), what in return surely gives different results.

A third cause might be the solver settings that we used. We used the highest available accuracy regarding the turbulence model, whereas the previous master thesis [4] was based on the 1st order option for the advective scheme in the turbulence model. Since these solvers are very sensitive to the selected model, it is plausible that this is the main reason for the lower values.

There is still the need of a comparison of reliable experimental data in order to validate the CFD model. However, at the time of making the present study, laboratory data was not available.

7 Conclusions

The two-step optimisation when using the method of Taguchi did not work out the way we hoped. This might be a result of a violation of the rules for orthogonality, since the statistical independency of the parameters was not fully guaranteed. This does however not mean that the method of Taguchi is unfit for an optimisation study of a horizontal axis impulse type turbine. By working with a fixed value for the relevant parameters, it is possible to exclude them at first for the optimisation. After the first results one could take the trial and error approach to further improve the configuration.

It is also clear that the waterjet should impact the blades under an angle of 90° as much as possible. Whether the best configuration is an angle of the inlet tube of 0 or -5 degrees combined with a blade inclination of 170º, 175º or 180º remains to be seen.

From the trial and error approach applied after observing the outcome of Taguchi method, it can be argued that the inlet tube angle is a crucial factor. Having done the tests for several turning velocities when changing the inlet tube angle, we could clearly see a big drop in power output after we decreased the angle of the inlet tube to 0 degrees.

The configuration of our best result consists of 22 blades, with a blade angle of inclination equal to 170º, a horizontal tube distance of 93.25 mm, a relative height of 183.25 mm, an angle of inclination of the inlet tube of -5° and a turning velocity of 270 rpm, producing a power output equal to 188 W and an efficiency equal to 36.7%. In comparison, classical vertical axis waterwheels have efficiencies on the order of 27%.

If we compare the configuration of our best result with the maximum power point generated in earlier research we find some discrepancies in the geometrical configuration. The previous work concluded that the highest power point was reached when the height of the nozzle was at a relative height of 169.45 mm (absolute height = 634 mm), while in this thesis the best results were found around a higher position, namely 183.25 mm. This gives a strong argument that further research is possible to determine which configuration is the best, as it might not always be possible to have a pre-defined setup in a real environment.

8 Economical cost

License cost for 8 computers		
Total simulation time: 800 hours	Cost of license: $230\ \frac{€}{license}$	Simulation hours/year: $1800\ \frac{hours}{year}$
	$= €817{,}77$	

Computer costs		
Total simulation time: 800 hours	Cost of 8 pc's: $3000\ \frac{€}{year}$	Working hours/year: $1800\ \frac{hours}{year}$
	$= €1333{,}33$	

Engineer costs	
Price per hour: $30\ \frac{€}{\boldsymbol{hour}}$	Number of hours spent on thesis: 400 hours
$= €12.000$	

Total cost		
€817,77	€1333,33	€12.000
	$= €14.151{,}1$	

9 Bibliography

[1] Agarwal. K.S. (2006). *Re-energizing Watermills for Multipurpose Use and Improved Rural Livelihoods.* Mountain Research and Development, 26(2): 104-108.

[2] Peace G. S. (1992). *Taguchi Methods: A Hands-On Approach*. Addison-Wesley, p114-137.

[3] Cornelis S. (2016). *Parametric study of the performance of an impulse-type turbine with CFD.* Master Graduation Thesis. University of Girona, Department of Mechanical Engineering and Industrial Construction. P14-15.

[4] Chin Fei N., Mizamzul Mehat N., Kamarudddin S. (2013). *Practical applications of Taguchi method for optimization of processing parameters for plastic injection moulding: a retrospective review*. ISRN Industrial Engineering, ID 462174, 1-11.

[5] Wei-Hsin C., Shih-Rong H., Yu-Li L. (2015). *Performance analysis and optimum operation of a thermoelectric generator by Taguchi method.* Applied Energy158 (2015) 44-54.

Annexes

A. Dimensions of the turbine

Table 18: The most important dimensions of the laboratory water turbine. These were obtained during earlier research [3]

Box			Wheel plates		
Length	553	mm	Width	50	mm
Height	650	mm	Length	50	mm
Width	175	mm	Height	1,5	mm
			Number of plates	16	
			Angle	0	°
			Position of plate to wheel	15	mm
Plate			**Inlet tube**		
Diameter	418	mm	Inner diameter	20	mm
Space between two plates	50	mm	Length	73	mm
Height of center	280	mm	Position of center	496	mm
Width of center	87,5	mm	outer diameter	25,2	mm
Length of center	275	mm			

B. The CFD model

In this annex, you will find the necessary information regarding the water turbine in the ANSYS environment. As explained in the methodology, the simulations were done in the Fluid Flow CFX domain. To be able to do this, the geometry and mesh had to be defined, which was done based on the information in Table 18 in A1. Fig. 36 shows the schematic project in ANSYS Workbench.

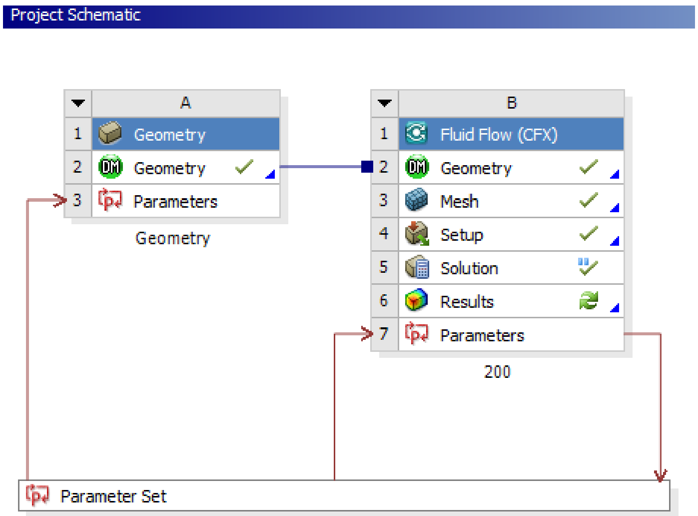


Figure 36: The main tree, with all the necessary steps to be completed until you get the results. Note that the blue line indicates that the geometry is imported.

B1 .Geometry

The goal was to make the geometry as close to the existing model as possible, without including elements that are not necessary for the simulation. It was thus decided that the geometry would only feature half of the turbine (in the longitudinal direction), using its symmetry. This was done to minimise the element size and calculation time, and it also means we should multiply the resulting power outputs by 2.

The geometry is split in different parts. The box, which represents the existing casing of the turbine, is one part, and its main function is collecting the water and redirecting the flow through the exit. A difference regarding the model versus the real turbine is the wider gap between the water wheel and the bottom of the box. This is necessary to avoid errors during the simulations. The turbine is the second part, one that is subtracted from the rotating area. This region includes every rotating part; hence we define the angular velocity for this region. The inlet tube is where the water enters the turbine. It is very important for calculating reasons to have a very accurate mesh in this region, so another separate body was made for this.

This leads to the geometry shown in Figure 37. A detailed description on how we made this will follow below.

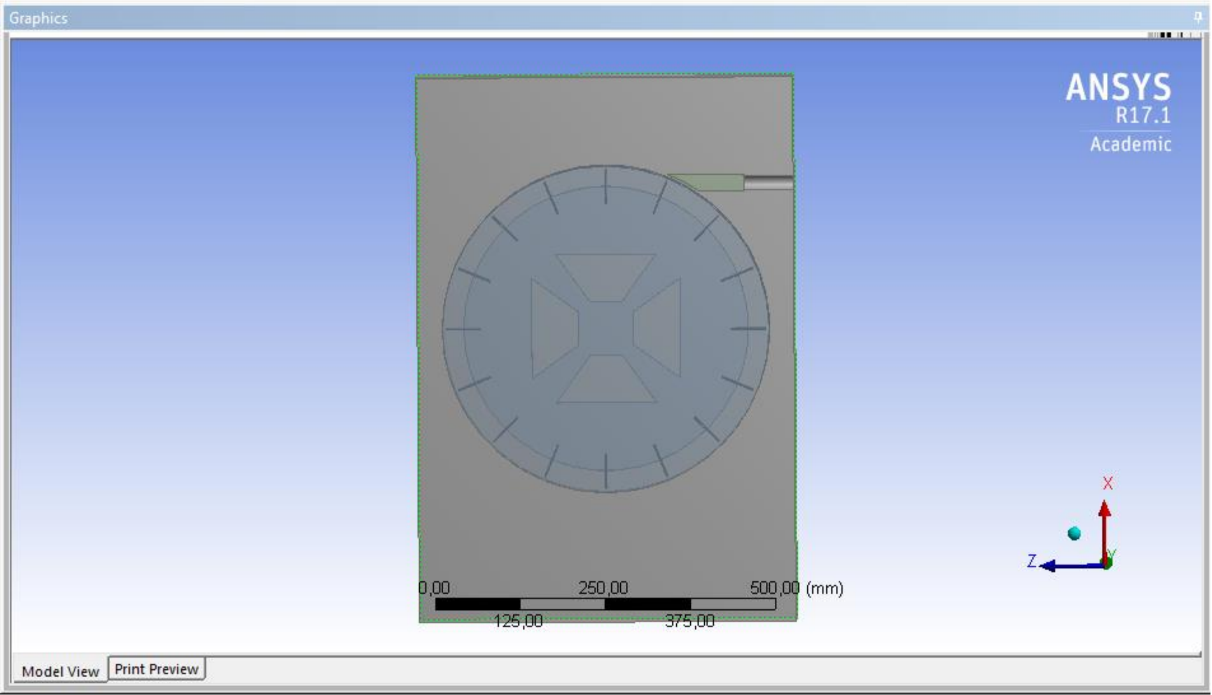


Figure 37: Geometry of the horizontal axis waterjet turbine. Note that only half of the turbine is created to save on simulation time and data space.

First the static part was made based on the real turbine's dimensions. Note that the values in Figure 38 are slightly higher than the real part (necessary to avoid errors), but these parts have no influence on the resulting power output.

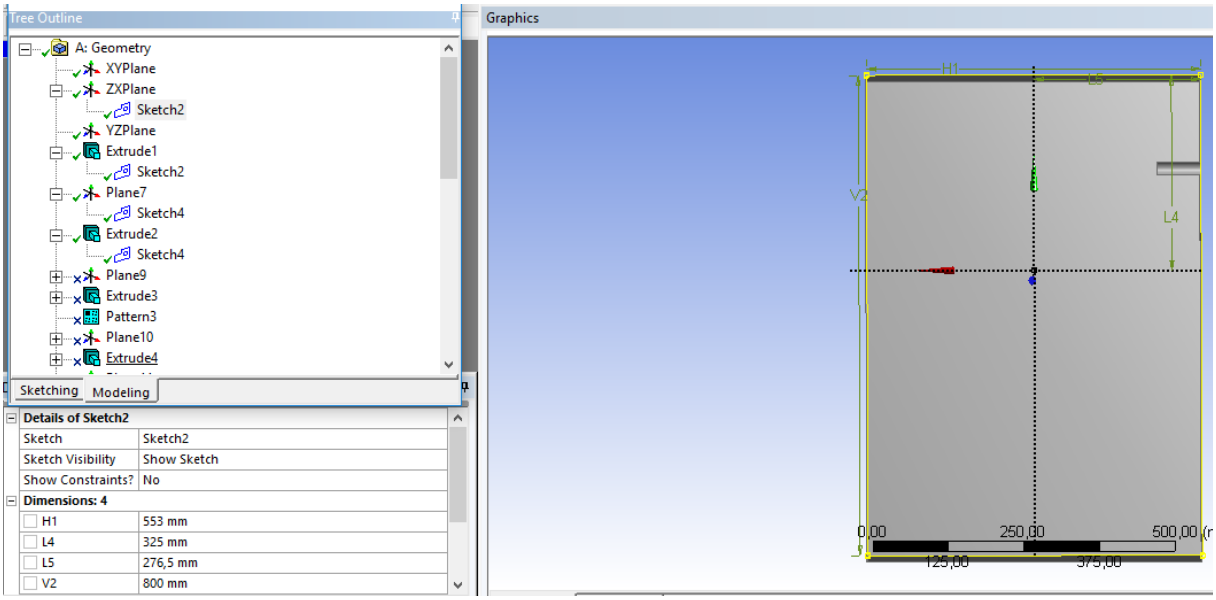


Figure 38: Extrusion 1. This shows the frame of the turbine. Also note the inlet tube, this was not created in this stage.

The first extrusion was used as a base for a new origin to define the water wheel, shown in Figure 39.

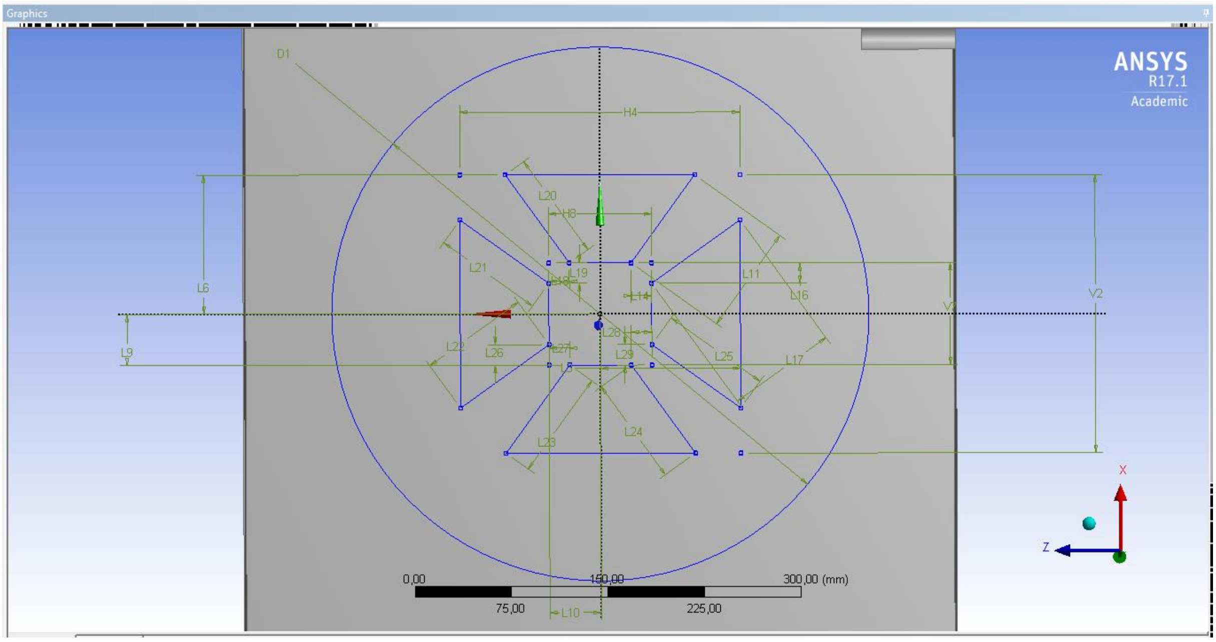
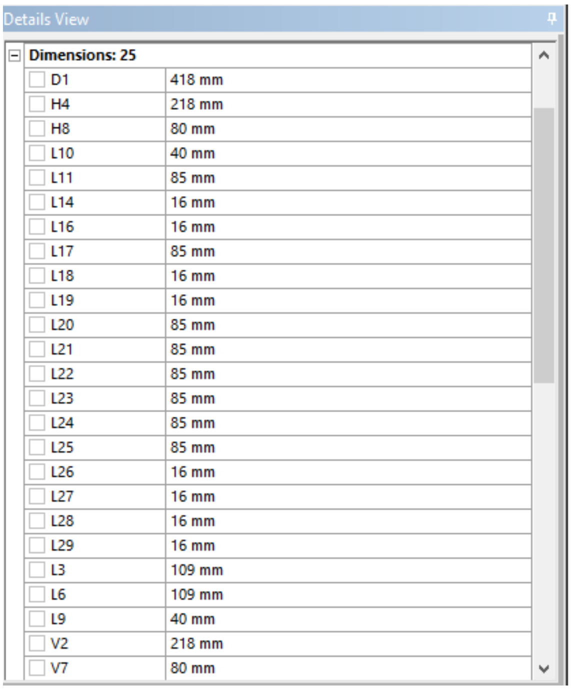


Figure 39: Sketch 4, where the water turbine is created.

The dimensions are shown in Figure 40



Details View

Dimensions: 25	
D1	418 mm
H4	218 mm
H8	80 mm
L10	40 mm
L11	85 mm
L14	16 mm
L16	16 mm
L17	85 mm
L18	16 mm
L19	16 mm
L20	85 mm
L21	85 mm
L22	85 mm
L23	85 mm
L24	85 mm
L25	85 mm
L26	16 mm
L27	16 mm
L28	16 mm
L29	16 mm
L3	109 mm
L6	109 mm
L9	40 mm
V2	218 mm
V7	80 mm

Figure 40: Dimensions of Figure 39

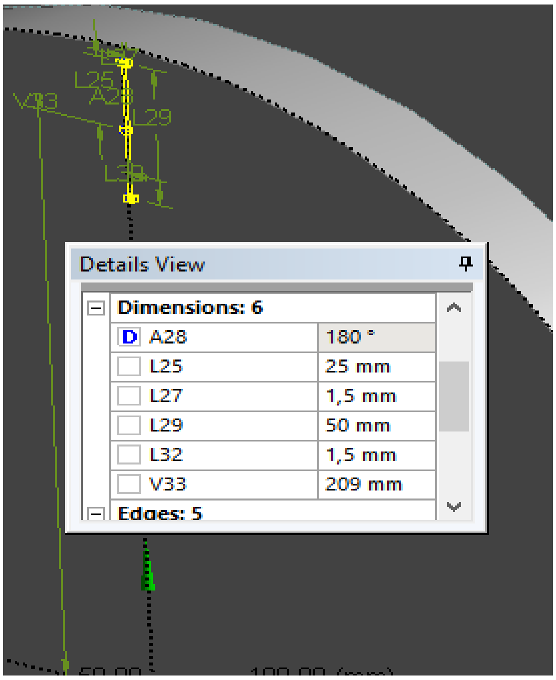
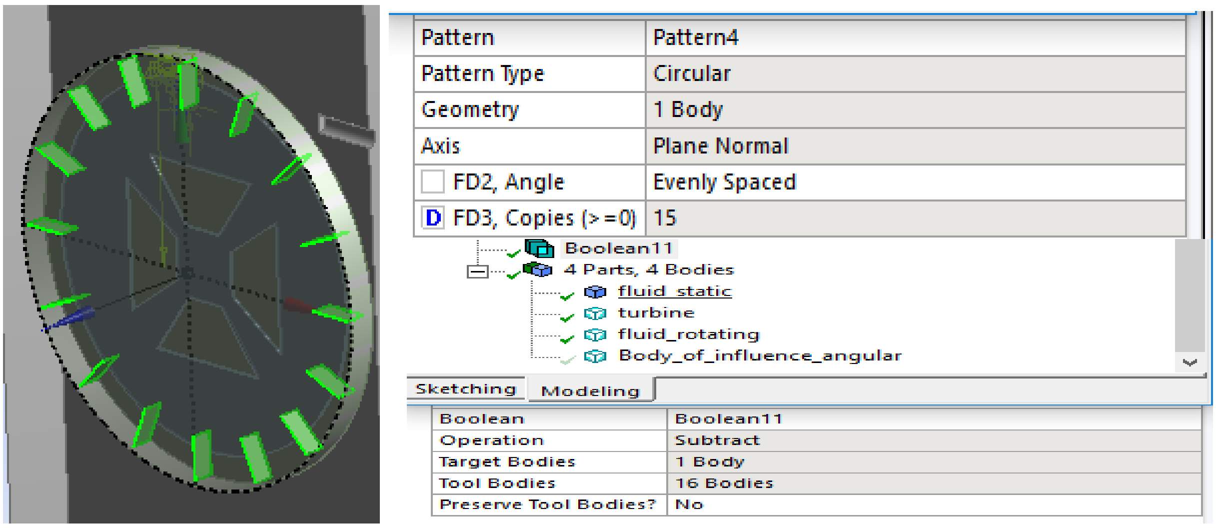


Figure 41: Construction of the blade

Then, the construction and parametrisation of the blades' angle was done, as seen in Fig. 41. The number of blades were generated by using the pattern command as shown in Figure 42.



Pattern	Pattern4
Pattern Type	Circular
Geometry	1 Body
Axis	Plane Normal
FD2, Angle	Evenly Spaced
D FD3, Copies (>=0)	15

Boolean	Boolean11
Operation	Subtract
Target Bodies	1 Body
Tool Bodies	16 Bodies
Preserve Tool Bodies?	No

Figure 42: Left: snap of the turbine with 16 highlighted blades. Right: shows that there are 16 bodies selected.

Note that the software sees the original and the added blades as different entities, so if we want to generate 16 blades we must give '15' as input for the 'number of blades' parameter.

In the extrusion for the blades (24,65mm) the option 'add frozen' was selected. This means that material is created, but it is added to the model as a frozen body, without merging them with other bodies in the model. Therefore, we should create a Boolean to merge them.

Next the inlet tube is created, it is already shown on Figure 39 and 42. To make sure we can apply an inclination to it, we must first define a new plane, Plane 16. This also enables us to parameterise its height. After this we sketch the feature (73mm x 10.35mm). The next step, related to the inclined geometry, is to add another plane, Plane 17. This is done to be able to use the feature ‘sketch instance’, where we can parameterise the ‘Rotate angle’, as shown in Figure 43:

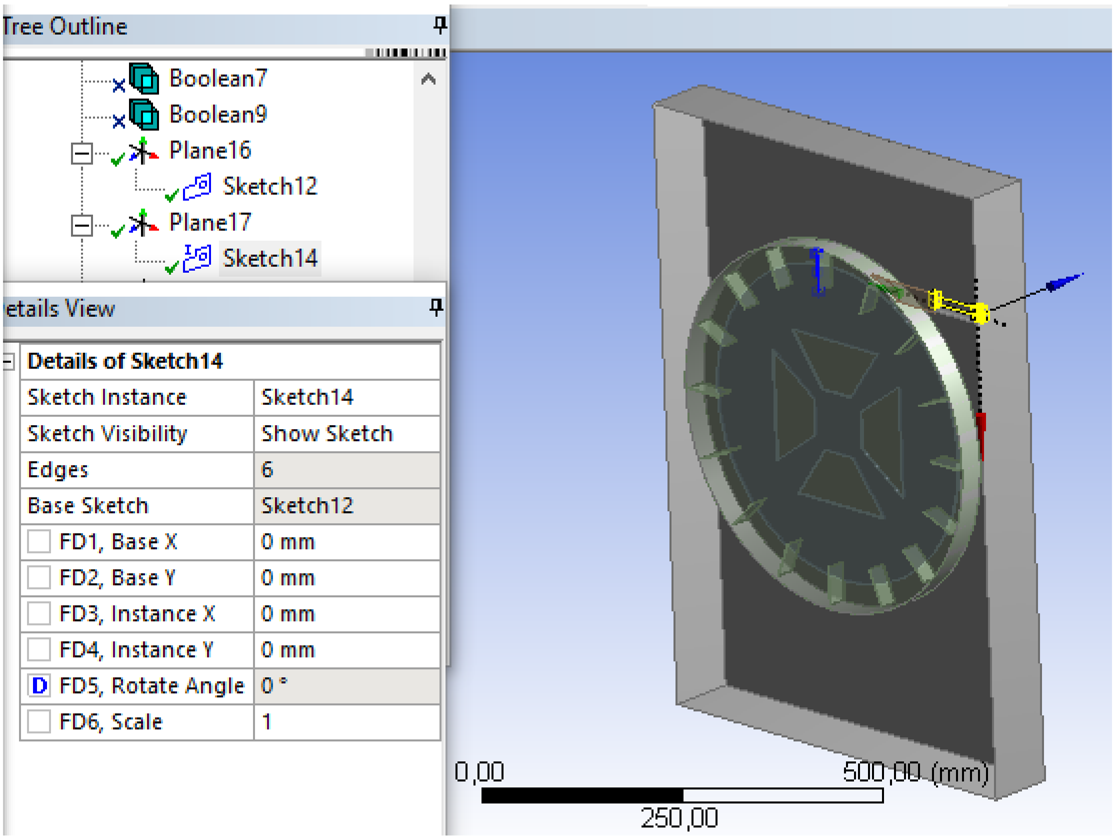


Figure 43: Parametrisation of the angle of the inlet tube.

To have a very fine mesh to represent the water flow, another part is modelled. We call it the ‘body of influence’ and is shown in detail on Figure 44, It can also be seen on Figure 45. It is constructed by making use of a circle and the trim functions.

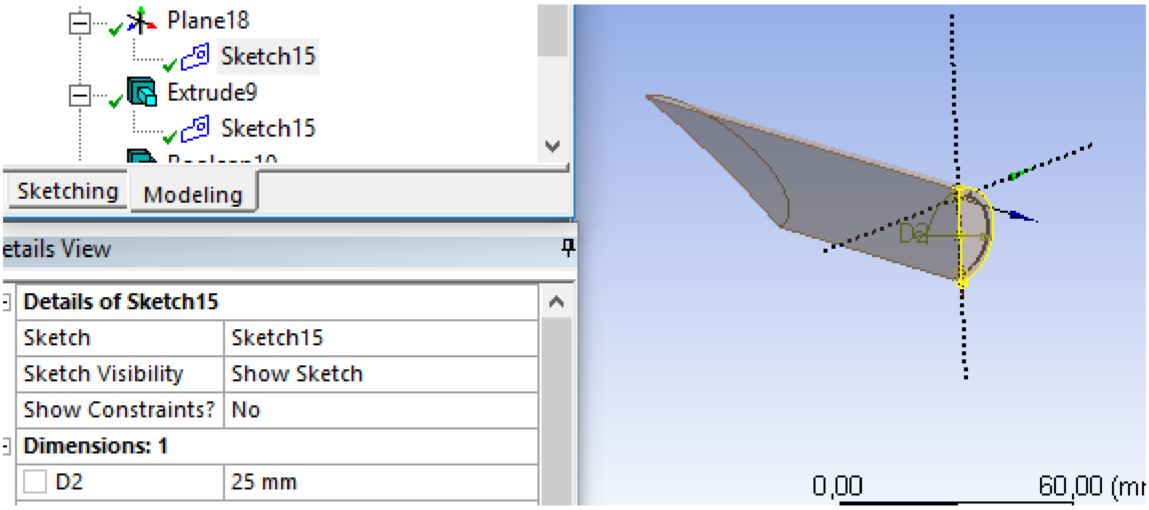


Figure 44: Creation of the body of influence

Afterwards, the rotating part has to be finished. This is done by defining two circles, 7 and 9, with an offset of 10mm (this is necessary to avoid errors during the simulations) and extruding them as shown in Figure 45. By using the Boolean function once more, we can merge them with the other parts in the geometry and cut off the Body of influence. It should be pointed out that we created a negative of the actual rotating part, but by using two more Boolean functions (6 and 9), this is of no concern for the simulations.

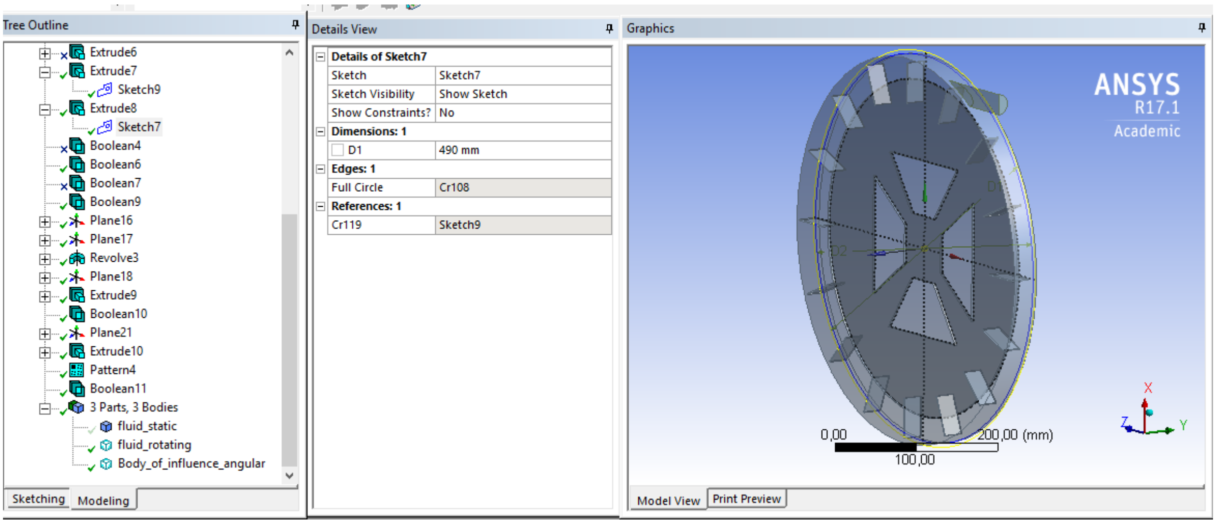


Figure 45: The finished rotating part. With the body of influence on the right.

Figure 46 on the next page shows the complete model.

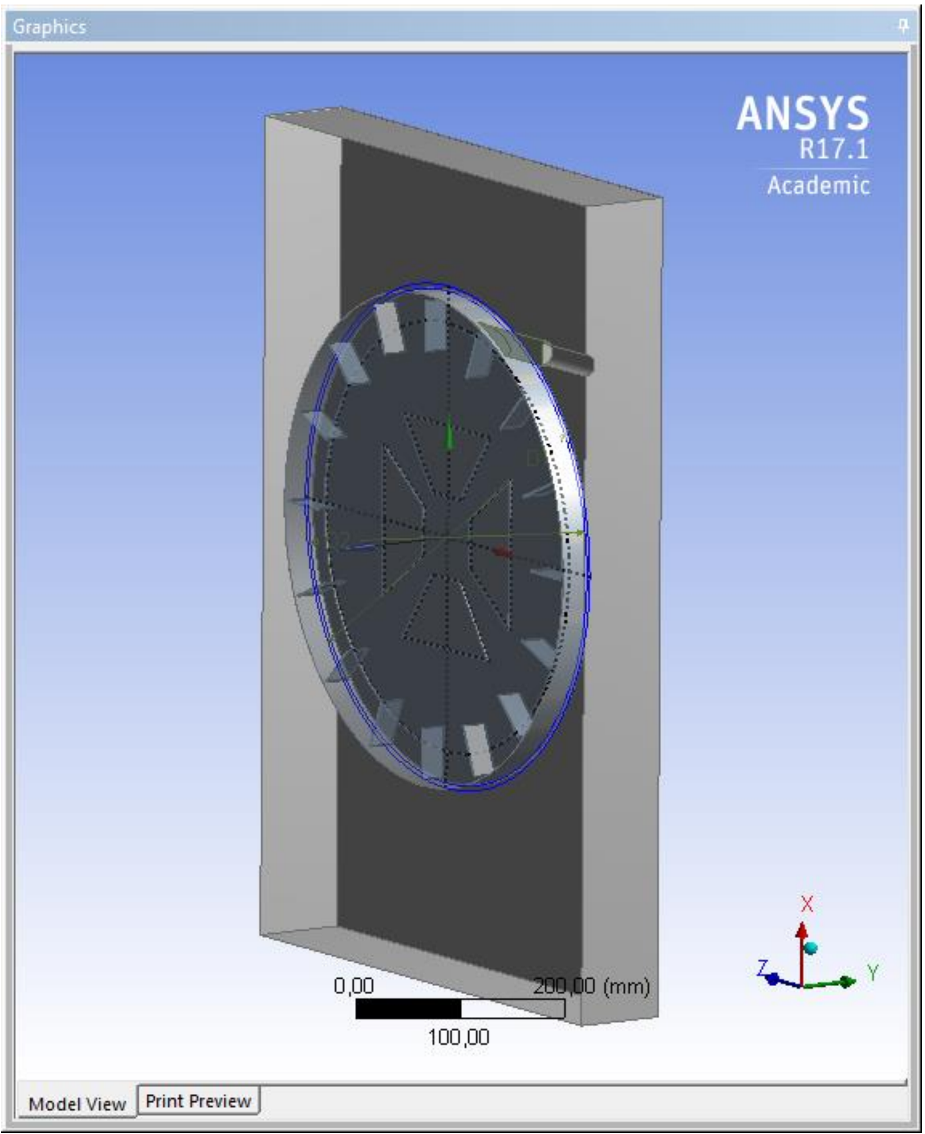


Figure 46: Example of a completely modelled configuration.

B2. Mesh

As pointed out in the text, it is necessary to have a fine mesh in specific regions and we must also take the limit of 512.000 elements into account. We could in theory define a different mesh according to the number of blades. This would however create another difference to consider while processing the results, and would make the exercise too difficult. As a result, we made the mesh for the case of 22 blades and used this one as the template for every simulation.

To be able to define different kinds of meshes for the various parts of the geometry, first several named selections are created, as shown in Figure 47. The names imply whether the selected part will rotate or remain static.

Figure 47: Left: the different parts of the model. All under a name so we can apply a customised mesh. Right: The static Left part highlighted.

The general settings for the meshing are shown in Figure 48.

The size function is set on ‘uniform’, even though the option ‘proximity and curvature’ fits the geometry better. This would however result in an amount of elements that easily exceeds three million, so it does not fit the boundaries of the used software. The smoothing is high, this is done to reduce the skewness and thus having a better mesh quality. The growth rate is set to thirty, again a measure to avoid the exceeding the limit of elements. It means that every element will be maximum 30% bigger than its neighbour.

The majority of parts have the same settings for the mesh. Exceptions are the parts that are essential to the calculations, so they need a much more refined mesh. This is done by using the Face sizing function. The Rotating blade edges, the static tube inlet, the Rotating Blades and the Static Tube Lateral have a respective element size of 1.5, 2, 3 and 30 mm. The Body of Influence has an element size of 3mm, and the mesh is produced by a body sizing function. The last more refined mesh is the rotating part as awhole. This is done by using the function inflation. What all those exceptions have in common is the lower growth rate of 20%. An example is given in Figure 49.

Details of "Mesh"

Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	CFX
Relevance	0
Shape Checking	CFD
Element Midside Nodes	Dropped
Sizing	
Size Function	Uniform
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Min Size	Default (0,48670 mm)
Max Face Size	50,0 mm
Max Tet Size	50,0 mm
Growth Rate	1,30
Automatic Mesh Base...	On
Defeaturing Tolera...	Default (0,243350 mm)
Max Dual Layers in Thi...	No
Minimum Edge Length	1,50 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,77
Maximum Layers	4
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Number of CPUs for P...	Program Controlled
Straight Sided Elements	
Number of Retries	0
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled

Figure 48: General settings for the meshes.

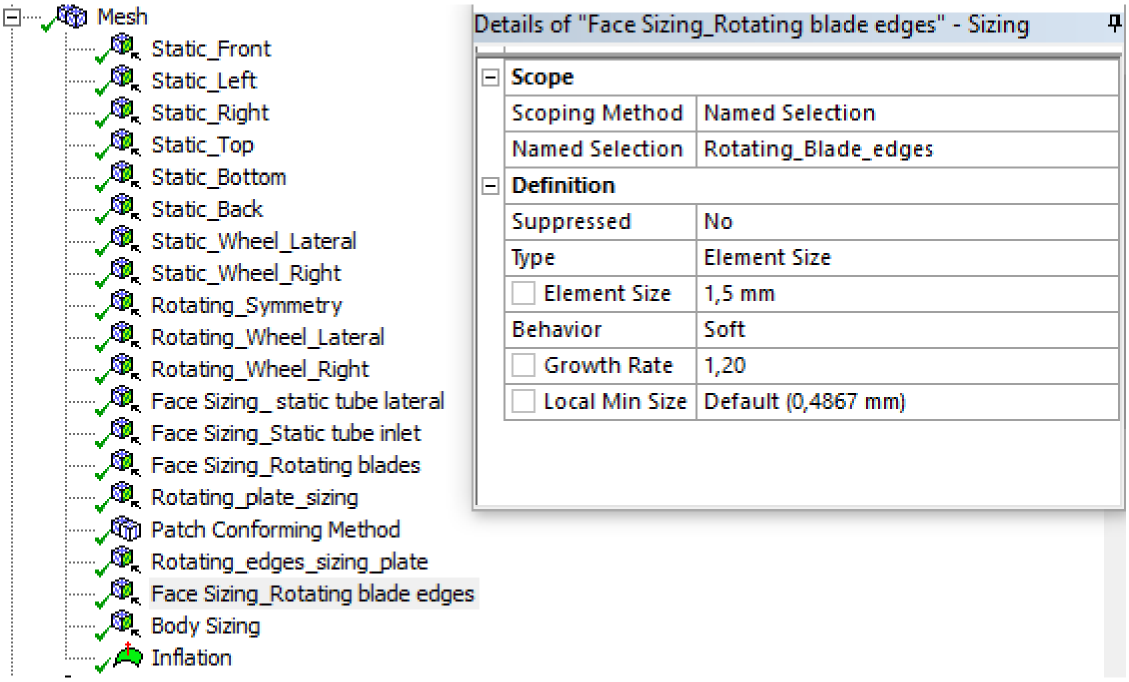


Figure 49: Customised mesh settings.

Figure 50 shows the mesh of run 10 from Table 9.

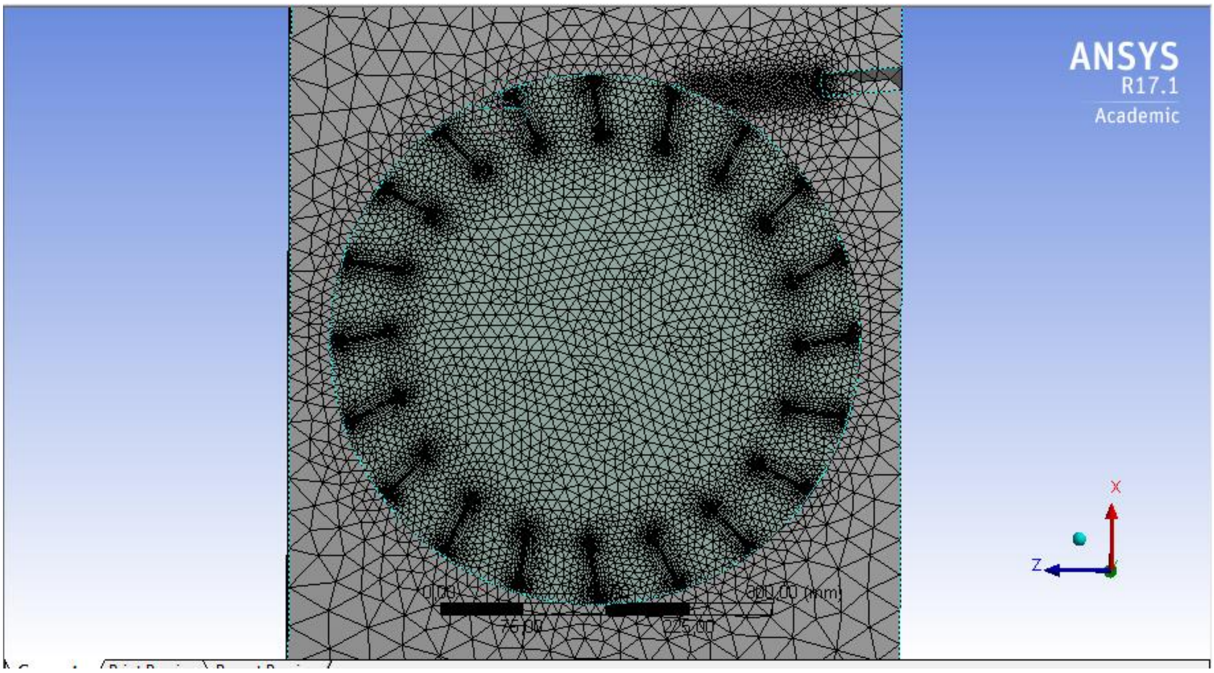


Figure 50: Mesh from run 10 from table 9.

B3. Setup

As mentioned in the thesis, the setup is done in the CFX – Pre-software program. Here we can define all the circumstances that might influence the simulations, for example the gravity. Figure 51 shows the general overview of the options in this window.

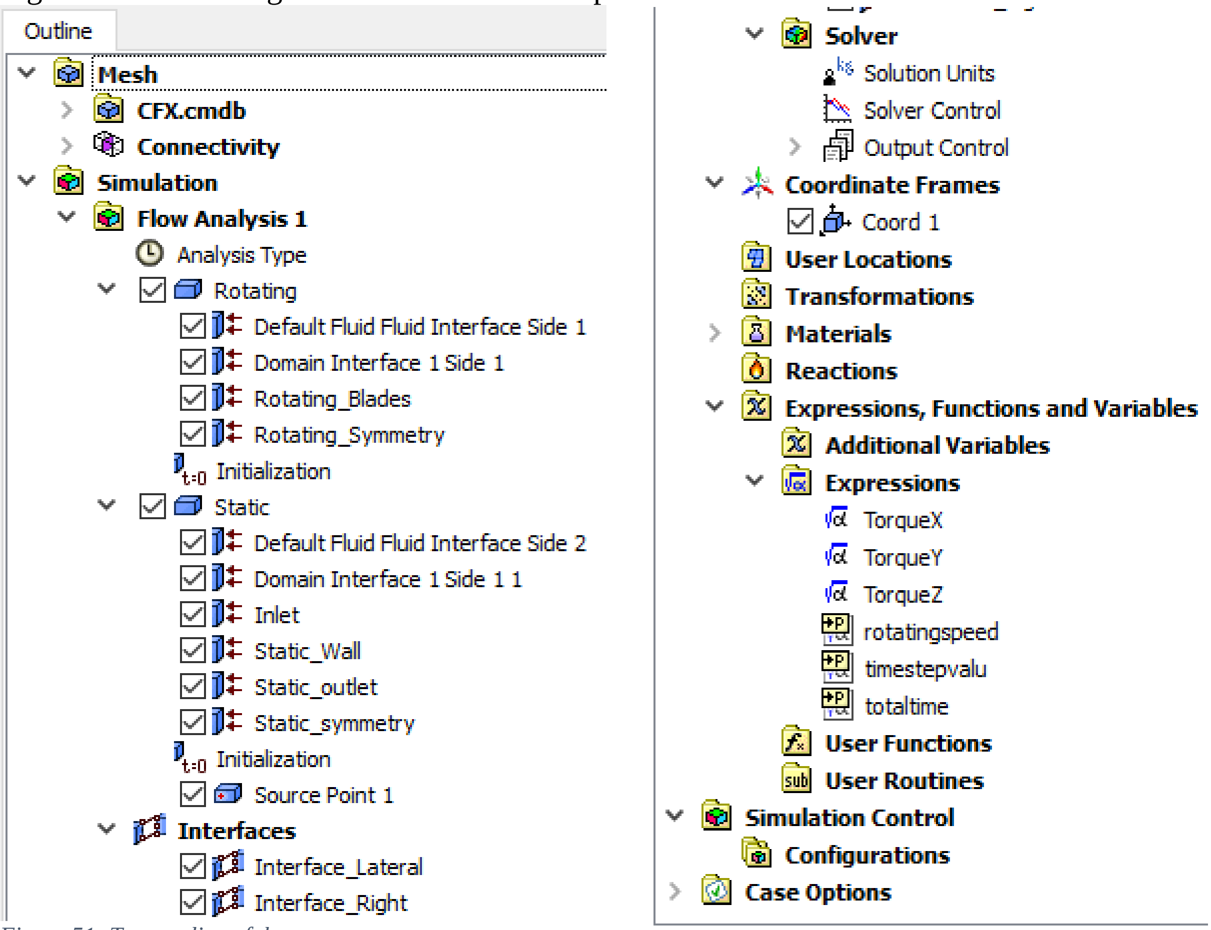


Figure 51: Tree outline of the setup.

The explanation of all relevant features has been done in Chapter 5.3. What we will describe here is how to make the expressions. Which is relatively straightforward.

First you insert an expression by clicking on ‘Expressions’ in the ‘Tree Outline’. There you can define the name, which is the one we can see on the image. The visible code in Figure 52 is a CEL code, and means that the torque around the x – axis of the pre-defined coordinate system ‘Rotating_Blades’ can be obtained. The same can be done for the torque around the y - and z – axis. To confirm the code, press ‘apply’.

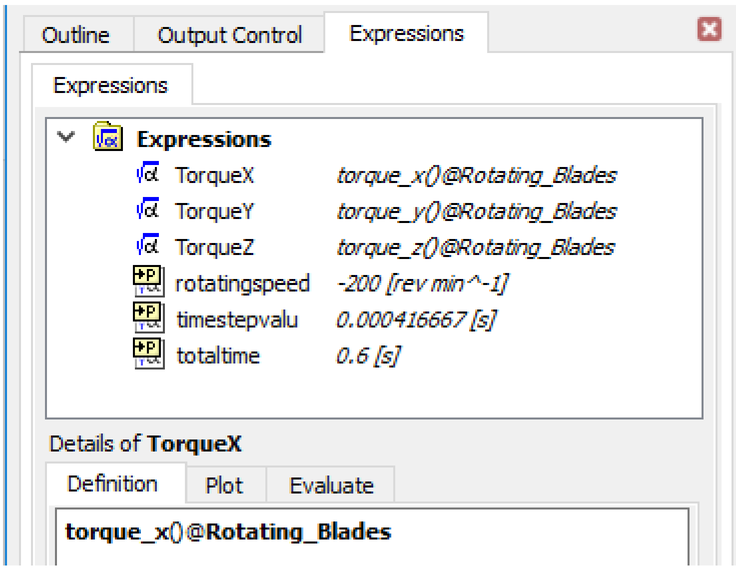


Figure 52: Shows the expressions to calculate the Torque.