Implementation of the effect of turbines on water currents in MOHID Modelling System

Óscar Balsells Badia

Thesis to obtain the Master of Science Degree in

Environmental Engineering

Supervisor: Prof./Dr. Ramiro Joaquim De Jesus Neves

Examination Committee

Chairperson: Prof./Dra. Ana Fonseca Galvão

Members of the Committee: Prof./Dr. António José Nunes de Almeida Sarmento

Prof./Dra. Lígia Laximi Machado de Amorim Pinto

September 2017

ABSTRACT

The present document describes the project of including into the hydrodynamic model of MOHID Studio application the effect of turbines on water currents. This implementation intends to be a reliable tool for promoting renewable energies that make use of hydrokinetic energy, as tidal stream energy. This model allows users to study flow modification and energy extraction as well as other sub-consequent effects as sediment transportation and water level.

The implementation is designed for axial turbines that can work on both directions, with free rotation on the vertical axis and pitch control for the power harvested. It can work in 2D and 3D simulations, even though the turbine is only discretized in the vertical direction (for 3D simulations only), not in the horizontal plane. The horizontal resolution, in order to obtain useful results, must be as little as the dimension of the turbines blades. In low resolution of the grid the application works well but the results are graphically inaccurate.

The programming is made in Fortran90, the same language as the MOHID Studio application. The code, which is not included in this memory, can be found in MOHID github repository [13]

Keywords: **[Tidal energy] [Current Energy] [MOHID Water] [MOHID Studio] [Turbine] [Finite Elements]**

ACKNOWLEDGEMENTS

Thanks to all the department of MARETEC for the help during the development of this thesis. Especial thanks to Ramiro Neves for tutoring the project and giving advice in the development, to Ricardo Miranda for the assistance and teachings in the programming and to Guillherme Franz and João Sobrinho for the help in MOHID as a user.

INDEX

ABSTRACT **II**

ACKNOWLEDGEMENTS **III**

INDEX **IV**

LIST OF TABLES **V**

LIST OF FIGURES **VI**

1. INTRODUCTION **1**

1.1. GENERAL OVERVIEW 1
1.2. CURRENT ENERGY 4
1.3. MOHID WATER 6
1.4. OBJECTIVE 8
1.5. SCOPE OF THE PROJECT 8

2. DEVELOPMENT **9**

2.1. HYDRODYNAMIC MODEL 9
2.2. DESCRIPTION OF THE IMPLEMENTATION 11
2.2.1. Design considerations 11
2.2.2. Model fundamentals 11
2.2.3. Discretisation 15
2.2.4. Output data 18

3. RESULTS **19**

3.1. GENERAL SETUP 20
3.2. SIMULATIONS 21
3.2.1. Horizontal channel 21
3.2.2. Horizontal channel 2D 25
3.2.3. Diagonal channel 27
3.2.4. Array layout 29
3.2.5. Real case 32

4. CONCLUSION **34**

4.1. ACHIEVEMENTS 34
4.2. LIMITATIONS 34
4.3. FUTURE WORK 35
4.3.1. Verification of the implementation 35
4.3.2. Improvement of the implementation 35

5. REFERENCES **38**

APPENDIX A **40**

A.1. INPUT DATA 40
A.2. MOHID CONFIGURATION FILES 42

LIST OF TABLES

Table 1. List of simulations. Own source 19

Table 2. Basic parameters values. Own source 20

Table 3. Input Data keywords. Own source 40

LIST OF FIGURES

Figure 1. Total primary energy supply by fuel 1
Figure 2. World gross electricity production (%) by source............ 2
Figure 3. World total gross of electricity production............ 2
Figure 4. Tide range all over the world............ 5
Figure 5. MOHID graphical representation............ 7
Figure 6.Arakawa C manner 10
Figure 7. Power coefficient and power extraction evolution with current speed............ 14
Figure 8. Vertical discretisation of the turbine area 15
Figure 9. Option 1 for computing the velocity modulus in the cell............ 17
Figure 10. Option 2 for computing the velocity modulus in the cell............ 17
Figure 11 Velocity modulus in the horizontal plane in the layer 12 21
Figure 12. Vertical cut in the x axis of the flow field............ 21
Figure 13. Vertical cut in the y axis, frontal view of the turbine............ 22
Figure 14. Effect of the turbine in the water level. 23
Figure 15. X-Y Graph of the velocity, power and energy output. 23
Figure 16. Velocity modulus in the horizontal plane. 25
Figure 17. X-Y Graph of the velocity, power and energy output. 26
Figure 18. Velocity field in the layer 12 27
Figure 19. X-Y Graph of the velocity, power and energy output. 28
Figure 20. Velocity field for the layout 2. 29
Figure 21. Velocity field for the layout 1. 29
Figure 22. Power and velocity data for the layout 2............ 30
Figure 23. Power and velocity data for the layout 1............ 30
Figure 24. Total power extraction of both arrays.. 31
Figure 25. Placement of the turbines in the Tagus Estuary............ 32
Figure 26. Power and velocity output of one group of 10 turbines............ 33
Figure 27. Energy and power output of the 40 turbines placed in the Tagus Estuary. . 33
Figure 28.Example of 2D discretization............ 36
Figure 29. Example of input data. 41
Figure 30. Example of hydrodynamic file with the implementation activated. 42
Figure 31. Example of Nomfich.dat file 42

1. INTRODUCTION

1.1. GENERAL OVERVIEW

To contextualize this project, a wider overview of the nowadays “energy problem” should be provided. Human activity, mainly the fact of burning fossil fuels for obtaining energy, is affecting the environment in a non-sustainable way: CO2 pollution is reaching record values, the mean global temperature has increased in almost 1 ºC since 1880 [1] and urban air-pollution is reaching unhealthy levels. These are just the main ones, the list is quite long; small variations can have terrible environmental consequences in the ecosystem and all the living species on the planet, including humans. Some changes have to be done in order to build a cleaner and more sustainable energy system able to deliver worldwide secure, affordable and sustainable energy.

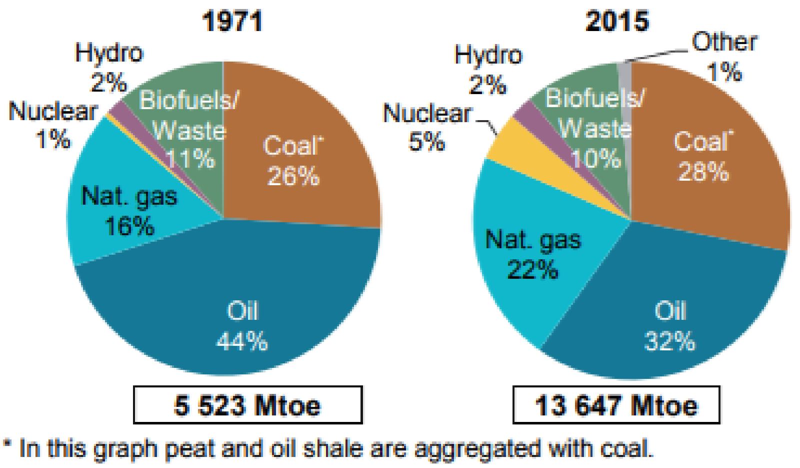

Figure 1. Total primary energy supply by fuel in 1971 and 2015. In other there are included renewable sources as geothermal, solar, wind, etc. Source: IEA [2]

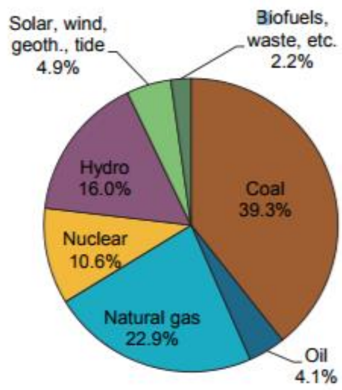

Figure 2. World gross electricity production (%) by source in the year 2015. Source: IEA [3]

According to the last data provided by the IEA (International Energy Agency), the primary energy supply all around the world comes mainly from fossil fuels (figure 1). The same happens with electricity production (figure 2). Taking into account the fact that the world population is expected to have grown up to 9 billion in 2050 and that emerging countries will continue grow technologically, the world energy demand is compelled to increase, following the actual tendencies as it is shown in figure 3.

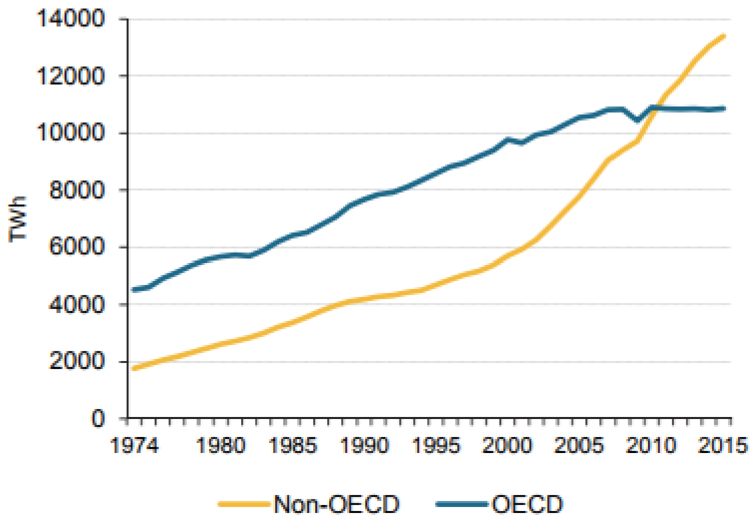

Figure 3. World total gross of electricity production of OECD and non-OECD countries. The OECD, Organisation for Economic Cooperation and Development, is composed by 35 states, including basically the world's most advanced countries. Source: IEA [3]

In this graph it is shown how the OECD countries gross energy production increase slightly in the last 5 years compared to the non-OECD countries. An interesting fact is that the OECD countries had experimented a small decrease in electricity production from fossil fuels [3] and an increase in renewable sources [4], so there is a willing to change the energy system towards a renewable one, but for the moment, is not sufficient. The actual energy system has an expiring date and need to change quickly. An energy sustainability between the environment, the social welfare and the economy is needed to procure a global energy system compatible with a sustainable development. The main issues are: equity in access and affordable energy services, environmental impact, greenhouse gas emissions and resource preservation. To achieve that, the next points should be take into account:

- Restrain the demand/consumption. Make a responsible use of energy and resources.
- Improve technological efficiency of production, transportation and consumption points.
- Increase the use of renewable energy technologies.

This project is thought to contribute to the development of renewable energies as marine ocean currents, tidal stream currents or river currents, by developing the model of a turbine and studying its effects in water. Renewable energies are the cornerstone for a sustainable future, and due to the increasing demand of electricity consumption in the years to come, the electricity industry is one of the main points to take action on. At the present time around 22-24% (figure 2) of the electricity comes from renewable sources, and it is estimated to grow in the years to come.

There are many kinds of renewable energies: biomass, eolic, geothermic, marine, tidal, solar, etc. Several of these renewable sources present the challenge of being weather dependant and cannot produce on demand, which difficult its development. Another handicap is that some of them are a little unpredictable, as wind energy. The electrical greed needs stable energy, and if it can be on demand, better. Current energy is the same as wind energy, it cannot be produced on demand, but tides and oceanic currents are more predictable than wind, and they can contribute as base energy in energetic systems.

The other big challenge is the cost. A lot of renewable technologies are actually in a prototype and development phase, and the economic conditions to make them profitable are too strict. For example, in the actual state of development of hydrokinetic energies, a high average velocity is needed to be economically profitable. This is where energy policies come in, since they have a huge influence on the development of renewable energies. Though they sometimes seem to forget about the economic profit and growth, with the proper development and research, they can be made more efficient and profitable.

It is the duty of the people, governments, research centres, companies, of everyone to boost renewable energy towards fossil fuels, in a sustainable way. This effort is necessary for a sustainable future, for the planet itself. Today energy companies, mainly with economical objectives, are the ones who rule the energy system, but this needs to change, energy lobbies need to be faced in detriment of a better quality of life and respect of the environment. The planet have enough renewable resources in order to provide the global energy needs, it only has to be done properly.

This thesis presents the following layout. First, it describes the current energy and introduce the software used and where the implementation is programed. Second, the physical concepts and mathematical formulation of the 3D hydrodynamic model that is used in MOHID. Third, the physical and mathematical formulation made for the current turbines is explained. Fourth, the model is applied on channels in different layouts to test the implementation and proof its reliability. Fifth and final, there are the conclusions and an explanation of the next steps in order to improve this implementation.

1.2. CURRENT ENERGY

The oceans represent a huge source of renewable energy. Nowadays, this energy is obtained through six different ways: waves, tidal range, tidal current, ocean current, ocean thermal energy and salinity gradient. The technologies in charge to take profit of this kind of energies, compared to other kind of renewable technologies, are at an early stage of development. In fact all ocean energy in general is in an early stage [5], with the exception of tidal barrage. There are then, two ways to produce energy from the currents in the ocean, tidal currents and marine currents. There is a third way of current energy, rivers.

The only difference of these three sources of water currents are the phenomena that origins them, but the power that can be extracted is the same in all of them, is only kinetic energy, and can be expressed as:

$$P = \frac{1}{2} A \rho U^3 \qquad \text{Eq. 1}$$

Where U is the velocity of the flow through the specific surface A. From this amount of energy only a portion can be extracted with turbines, which is modelled with a power coefficient. Tidal energy is the most developed [6] in the group of current energy extraction so a more detailed explanation is given.

Tides can be defined as the oscillatory motion of the ocean in which the mass of the ocean rises and falls alternately in a regular way. These oscillations are mainly due to the gravitational interaction of the Moon and the Sun on the Earth and the rotation of the Earth. These forces cause an alternation of potential energy which creates horizontal currents of water that we call rise-fall and flood-ebb currents. This currents and difference of water level can be used to generate electricity basically in two main procedures: either harnessing its potential energy using tidal barrages or its kinetic energy using stream devices, turbines. But in this thesis only current energy is the object of study. In figure 4 we can find the places where the biggest tidal currents can be found, taking into account that they appear near the coast caused by the narrowing of the geography, which causes high velocities with the rise and fall of the tides.

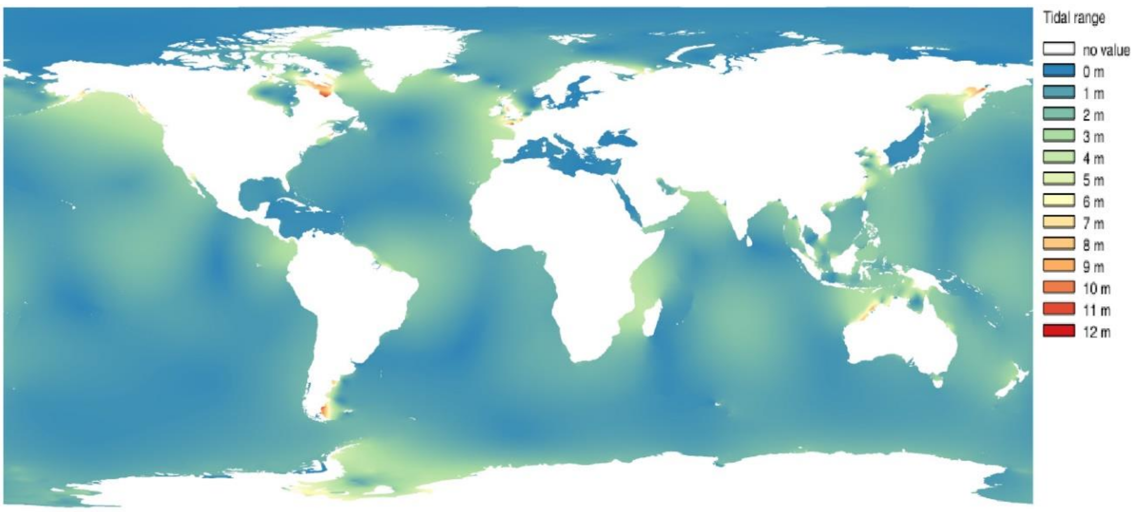

Figure 4. Tide range all over the world. This map shows the places with more tidal range and in consequence, they are candidates to have high velocity currents Source: https://www.ocean-energy-systems.org/ocean-energy-in-the-world/gis-map/

For marine currents, the only near-shore large-scale current swift enough to drive large electricity-generation are the subtropical surface western boundary currents [7], at the actual state of this technology development. For river currents, rivers with huge discharges as the Amazon are possible emplacements. Current turbines offer a less damaging alternative of the classical hydropower plants.

One of the strong points of this source, beyond its accessibility, is its predictability. Tides, for example, are more predictable than wind or sun, and they can be used in the future as base energy in the countries energy systems. Also, as the turbines work with in-stream currents, meaning that the current in which the turbine is placed is a natural one, the impact compared with hydropower plants or tidal barrage is smaller.

Even though current streams are renewable sources of energy, as all of the renewable energy technologies, they aren't environmentally friendly by definition [6]. All the activities involved in the manufacturing and maintenance have an impact on the environment. Their life cycle has an impact on the environment and further studies should be done in order to know their real impact. For example, the alteration in flow patterns cause a modification in the sediment transportation and in water level, and both effects have a direct impact on the environment. Also there is the environmental impact to the fauna habitat of the emplacement where the turbines will be placed, it can cause physical damage to the animals and also the noise caused by the turbines can disturb them. In definitive, even if they are renewable sources, they need to be implemented with caution and responsibility, watching all the possible effects.

1.3. MOHID Water

MOHID is an environmental modelling system dealing with transport and biogeochemical transformation processes in complexes geometries, developed at the Marine and Environmental Technology Research Center (MARETEC) at Instituto Superior Tecnioc (IST). It has multiple functionalities and can deal with multiple physical conditions. The actual MOHID model is able to deal with 1D, 2D and 3D simulations, Eulerian or Langrangian approaches and different vertical coordinates and cell geometries. It allows to run nested models in order to allow users to study local areas obtaining the boundary conditions from the father model [8].

It has two main cores, MOHID Land and **MOHID Water**, and can be used to simulate a wide range of processes as sediment transport, water quality, infiltrations, channel flows, etc. The implementation programmed in this thesis only affects the MOHID Water core, where the hydrodynamic model is programed.

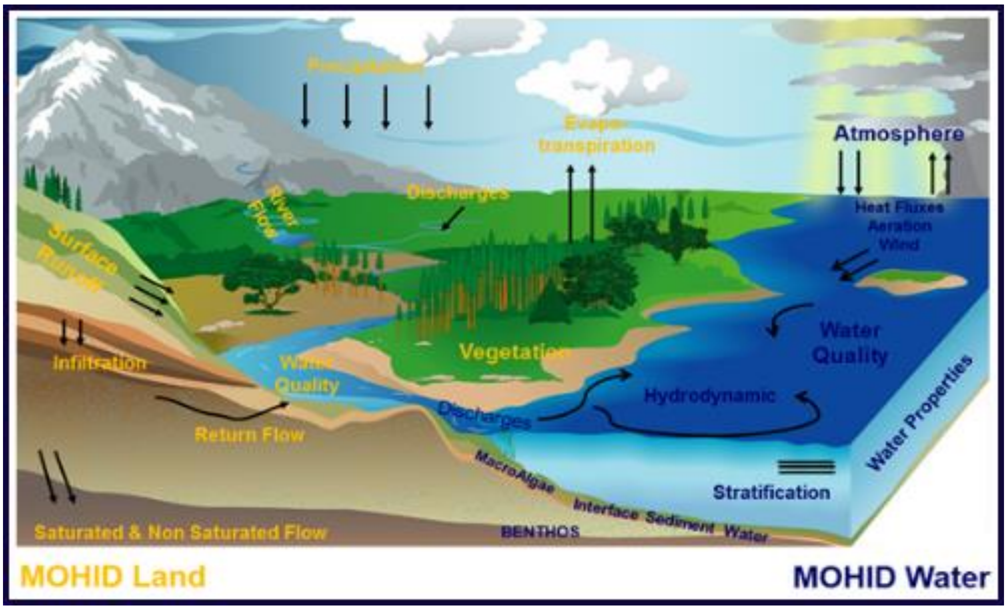

Figure 5. MOHID graphical representation. Source: http://www.actionmodulers.com

Nowadays, the whole model is programmed in ANSI FORTRAN 95 with an object oriented philosophy. It is a really complete model, with more than 40 modules and 150 thousand code lines. The code is open source with the idea to allow the inclusion of new developments, as it happens to be with this implementation.

In conclusion, MOHID is a really complete model that covers a long list of processes and has a huge quantity of implementations and capabilities. It has a lot of pre-processing and post processing tools, and deep knowledge of the processes involved is needed in order to make the simulation correctly. All this makes of MOHID a reliable decision support tool [9] which have been used for some coastal projects, and nowadays is the current working tool of MARETEC research centre.

1.4. OBJECTIVE

The main objective of the project is to implement in the MOHID hydrodynamic model the effect of extracting kinetic energy of water currents with turbines, providing users the possibility to see energy extraction and flow modification. The idea is to make a reliable tool that can be used by others in the MOHID application for any kind of studies that includes turbines and currents. As the project is Open Source, the implementation can be improved by other users in the future.

1.5. SCOPE OF THE PROJECT

The scope of the project can be summarised in the next goals:

- Give a brief introduction of the actual energy system and expose the idea of why current energy, tidal currents specifically, should be taken into account in the sustainable energy system of the future.
- Explanation of the implementation and all the simplifications considered.
- Program the implementation. The implantation should be programmed so it can be useful in further simulations, not only for this project.
- Verification of the implementation. It will be tested in different environments with MOHID Studio app and the results will be analysed.
- Weak points of the implementation and improvement.

2. DEVELOPMENT

2.1. HYDRODYNAMIC MODEL

The hydrodynamic model of MOHID solves the three-dimensional incompressible primitive equations. The Hydrostatic, Boussinesq and Reynolds' approximations are assumed in the equations presented [8]. All the equations here are written in the differential form with Cartesian coordinates.

The momentum balance equations, Navier-Stokes, for horizontal velocities are:

$$\frac{\partial u}{\partial t}+\frac{\partial (uu)}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z}-fv=-\frac{1}{\rho_0}\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(A_H\frac{\partial u}{\partial x}\right)$$
$$+\frac{\partial}{\partial y}\left(A_H\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(A_V\frac{\partial u}{\partial z}\right) \qquad \text{Eq. 2}$$

$$\frac{\partial v}{\partial t}+\frac{\partial (uv)}{\partial x}+\frac{\partial (vv)}{\partial y}+\frac{\partial (vw)}{\partial z}+fu=-\frac{1}{\rho_0}\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(A_H\frac{\partial v}{\partial x}\right)$$
$$+\frac{\partial}{\partial y}\left(A_H\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(A_V\frac{\partial v}{\partial z}\right) \qquad \text{Eq. 3}$$

The vertical momentum, if we assume hydrostatic pressure (neglecting vertical flow accelerations and diffusive transport), becomes:

$$\frac{\partial p}{\partial z}+\rho g=0 \qquad \text{Eq.4}$$

The continuity equation, assuming constant density becomes:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad \text{Eq.5}$$

The variables u, v and w represent the components of the velocity vector in the x,y and z directions respectively; f is the Coriolis parameter, A_H and A_V are the turbulent viscosities in the horizontal and vertical directions. The ρ_0 and ρ are the reference density and density respectively, and the p is the pressure.

The density is computed by the UNESCO equation of state as a function of the salinity, temperature and pressure. The turbulence is compute as a one-dimensional model, based on the GOTM model for the vertical and on empirical formulation for the horizontal.

In the model, there are two layers that differ from the rest: the bottom and the free surface layer. In the bottom, the shear stress can be computed with the assumption of a logarithmic velocity gradient and in the surface the shear stress from the wind can be also computed.

For the spatial discretisation MOHID uses a finite volume approach to discretize the equations. The discrete form of the governing equations is applied macroscopically to a cell control volume. It is interesting to highlight that the procedure of solving the equations is independent of the cell geometry, allowing almost all kind of shapes of the cell.

It is important to know that the grid is staggered in the horizontal in an Arakawa C manner [10]. For example, horizontal velocities are located in the centre-west (u-velocities) and south (v-velocities), while elevation is placed on the centre (figure 6). It is important for knowing where the calculated values and the interpolated ones are.

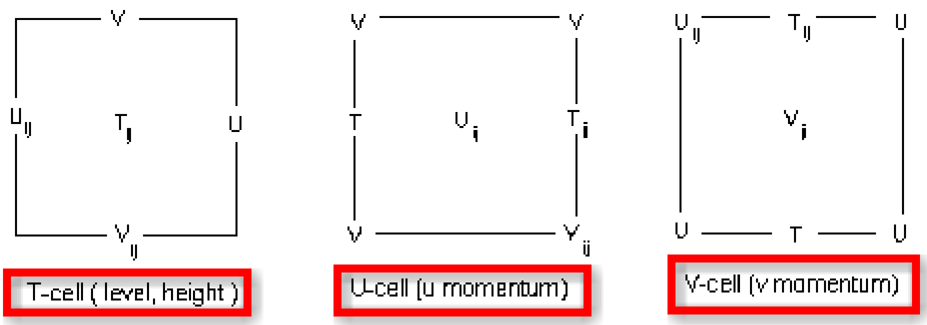

Figure 6.Arakawa C manner placement of the calculated parameter in a grid cell. Source: wki.mohid.com

The model allows different vertical coordinates as Sigma, Cartesian, Lagrangian, Fixed Spacing and Harmonic, been the Sigma and the Cartesian the more used ones.

Another important point is the temporal discretisation. It is done by a semi implicit ADI (Alternate Direction Implicit) algorithm. This algorithm computes each velocity component alternatively implicitly and explicitly. This allows preserving the stability advantages of implicit methods without the drawbacks of computational expensiveness and associated phase errors. Is it possible to choose between two different discretization [8], the Abbot scheme and the Leendertsee scheme.

2.2. DESCRIPTION OF THE IMPLEMENTATION

2.2.1. Design considerations

The most desirable approach to numerically model the impact of current turbines in the water flow for energy extraction would be to employ a full three-dimensional hydrodynamic model with an accurate representation of the flow-structure interactions between the current flow and the turbine. An accurate representation means to work with high resolution in the discretisation of the domain (vertical and horizontal) where the turbines are located. This implies a high resolution in the turbine geometry. Also, it should take into account that a part of the geometry rotates in the vertical plane (the blades of the turbine for axial turbines) and also can rotate in the horizontal plane, to adapt the direction of the turbine to the flow and maintain the perpendicularity between them. This implementation, at first sight is quite difficult to make by its own. There is also a problem in the relative scales of the processes involved. To model the turbine interaction with the flow, a small size of the mesh is needed, whereas for modelling tidal flow processes the resolution is a lot smaller (big size of cells). We have also the same issue with the temporal discretisation.

The idea then, is to provide the best approach possible to model the tidal energy harvesting and the impact of turbines in the flow. It’s important to find the balance between the spatial discretisation and the implementation in order to be able to join the processes of, for example, tides and turbines in the same simulation and have useful numerical and visual results. The implementation will be a 2D and 3D model in order to take profit of the 3D hydrodynamic model of MOHID explained in the point before and make the implementation more flexible in future simulations that can include this turbine approach. Working only in 2D will be a huge limitation for the potential of MOHID model.

So, the main design considerations are a 2D-3D model, in order model axial turbines that can turbine in both directions and with free rotation in the vertical axis.

2.2.2. Model fundamentals

There are several approaches to model the effects of a turbine in a 2D or 3D hydrodynamic model. Most of them are based on the same premise, which is to represent the turbine as a momentum sink by adding a reaction force (F_T) into the hydrodynamic model.

$$P_T = F_T U \qquad \text{Eq.6}$$

The power of the turbine can be interpreted as the product of the reaction force and the stream velocity (equation 6). The power of the turbine is an input parameter, and the velocity is given by the hydrodynamic model, so it is possible to calculate this reaction force and include it in the model to proceed with the simulation. Nevertheless, the equation 6 is not a good approach to implement the effects of the turbine into the flow, is just explicative.

To simulate the turbine, two main parameters need to be introduced [11]:

- The thrust force produced by the turbine rotor due to energy extraction, eq. 7.
- The power extracted by the turbine, eq. 8.

$$F_T = \frac{1}{2} \rho A_T C_T U^2 \qquad \text{Eq.7}$$

$$P_T = \frac{1}{2} \rho A_T C_p U^3 \qquad \text{Eq.8}$$

The ρ is the density of the sea water, the A_T is the area swept by the blades, C_T is the thrust coefficient that quantifies the force exerted by the turbine to the flow and C_P is the power coefficient that quantifies the amount of power extracted from the flow. The drag force exerted by the structure of the turbine is not contemplated in this model. Some models use the same coefficient for thrust and power, what means that the work done against the flow is the same as the energy extracted from it but, as is expected, thrust coefficient have to be greater than power coefficient. The power coefficient describes only the amount of power transferred to the rotor of the turbine as a torque, while the thrust coefficient is related to all the losses of kinetic energy of the flow, including the energy transformed into turbulent kinetic energy. This is why the power coefficient should be lower than the thrust coefficient. Also, it depends on the hypothesis and simplifications made for the calculation of both coefficients, which is also supported by experimental data [12].

$$C_T = \begin{cases} 0 & \text{sii} \quad U \leq U_C \\ C_{T0} & \text{sii} \quad U_C < U \leq U_D \\ C_{T0} \dfrac{U_D^3}{U^3} & \text{sii} \quad U > U_D \end{cases} \qquad \text{Eq.9}$$

For the power coefficient, the parameterisation is the same:

$$C_P = \begin{cases} 0 & \text{sii} \quad U \leq U_C \\ C_{P0} & \text{sii} \quad U_C < U \leq U_D \\ C_{P0} \dfrac{U_D^3}{U^3} & \text{sii} \quad U > U_D \end{cases} \qquad \text{Eq.10}$$

As both parametrisations of C_P and C_T are the same, the only difference between them lies on C_{P0} and C_{T0} constants which are the design values for both coefficients respectively. U_C and U_D are the cut-in and design speed. In figure 7 the parameterization of the power coefficient is plotted.

Both coefficients are programmed with a security factor of 15% of the velocity cut-in speed, avoiding oscillation values of the thrust force and power around the cut-in and design speed. When the turbine starts working, $U > U_C$, the reaction force may decrease the velocity again under the cut-in speed. With 15% of security factor, the turbine continues to produce energy and exert the thrust force till the velocity modulus U, decreases under $0.75 U_C$. Once it decreases below this value the turbine stops working and it has to increase again over U_C to start working again.

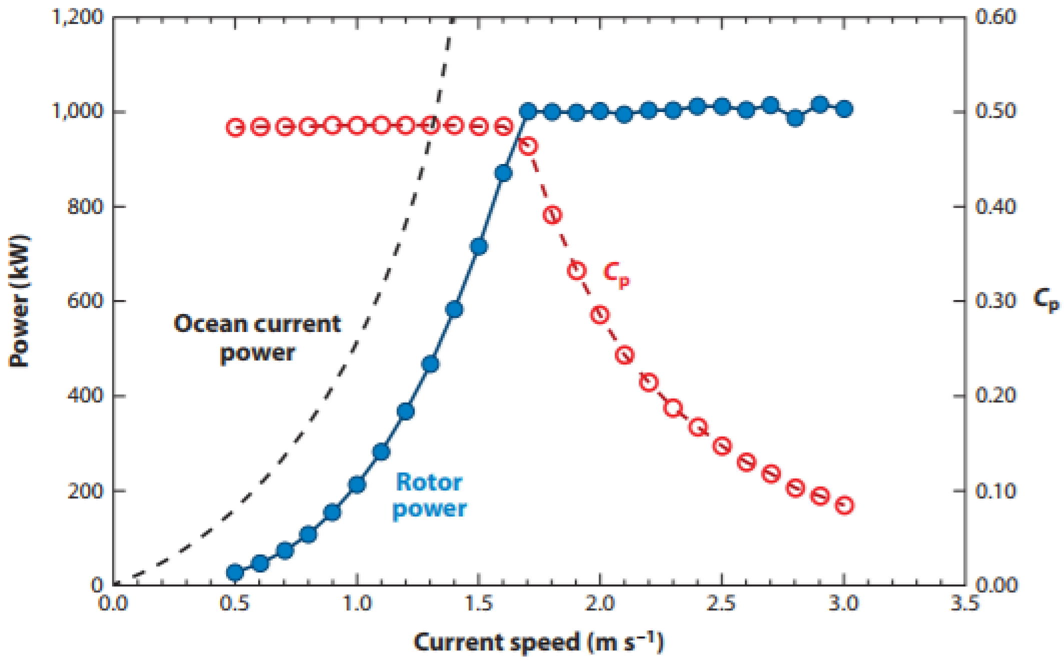

Figure 7. Power coefficient and power extraction evolution with current speed. Source: [7]

The energy extraction is calculated as the integration in time of the product of power and time differential. With a constant value of the time interval of the simulations, the equation for the energy is:

$$E_T = \sum_{i=0}^{n} P_T \cdot \Delta t \qquad \text{Eq.11}$$

Where n is the number of iterations and Δt is the time step of each iteration.

2.2.3. Discretisation

In this implementation, the spatial discretisation is only on the vertical direction. As a result, the force excreted by the turbine is a punctual force and the calculation of this force is made with the equation 7. The non-discretisation of the model in the horizontal axis simplifies quiet a lot the model and the perpendicularity between the turbine and the flow is implicitly assured.

$$A_{T_K} = \frac{r^2}{2}\theta - r \cdot \sin(\frac{\theta}{2}) \cdot d - \sum_{k=1}^{k} A_{T_{K-1}} \qquad \text{Eq.11}$$

The vertical discretisation is made through the equation 11, where A_{TK} represents the portion of the area swept by blades of the turbine that is in the layer k. In the figure 8 the equation 11 is represented graphically.

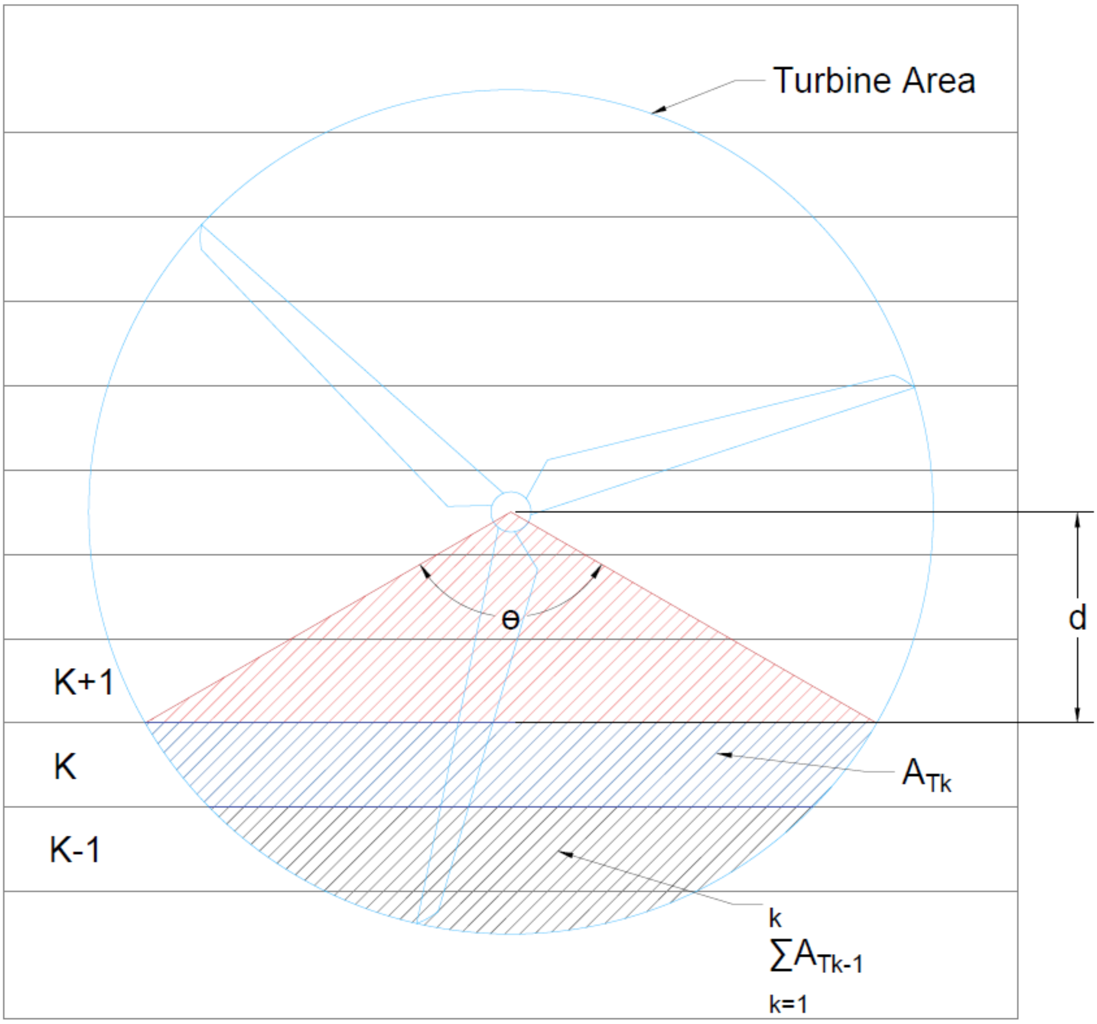

Figure 8. Vertical discretisation of the turbine area. Own source.

The force excreted by the turbine is calculated with the velocity of the flow in each layer while for the power, the velocity is an average value between the layers that contain the turbine. So, the equations 7 and 8 can be rewritten, for a turbine placed in the cell (I,J) coordinates, as:

$$\vec{F}_{T_K} = \frac{1}{2}\rho A_{T_k} C_T U_k \vec{U}_k \qquad \text{Eq.12}$$

$$P_T = \frac{1}{2}\rho A_T C_p U_{AV}^3 \qquad \text{Eq.13}$$

F_{TK} represents the thrust force made by the turbine on the flow in the layer k. $\vec{U}_k$ is the velocity vector of the turbine in an Arkawa C manner (the u component in the centre of the West face and the v component in the centre of the South face). U_k is the velocity modulus and U_{AV} is the average modulus velocity of the k layers of the cells in the coordinates i, j that contain the turbine, calculated as:

$$U_{AV} = \frac{\sum_K A_{T_K} * U_K}{\sum_K A_{T_K}} \qquad \text{Eq.14}$$

The last thing left to specify is where the velocity modulus U_K is calculated. There are two ways, both of them valid. The first one is to calculate it in the middle of the cell. Taking into account the Arakawa C grid distribution, the velocity modulus will be calculated with the velocities shown in the figure 9. This option makes that with the semi-implicit algorithm used in the model, the modulus value variations between u (velocity component in the x direction) and v (velocity component in the y direction) are minimal. Then, for calculating the thrust force in the nodes of the u and v velocity components (where the hydrodynamic model of MOHID calculates the velocities), we are using a velocity modulus not calculated in these points. The other one is to calculate the velocity modulus in the same point where the velocity components u and v are calculated, the centres of the West and South faces respectively. As can be appreciated in the figure 10, the modulus is calculated in each direction by the velocity components of the surrounding cells. With this option we are actually calculating (is an interpolated value also) the velocity modulus in the points where the velocities of the hydrodynamic model are computed. The issue is that we have different values of the velocity modulus when the model is computing the u component or the v component, as the calculations are made in different points.

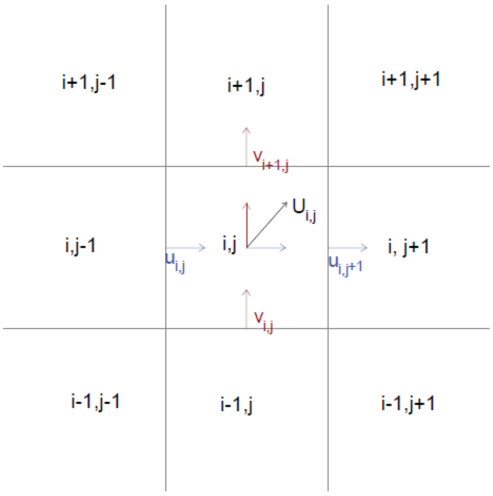

Figure 9. Option 1 for computing the velocity modulus in the cell. In this case the velocity modulus is calculated in the centre of the cell. Its value is the same when the model is computing one component of the horizontal velocity or the other. Own source

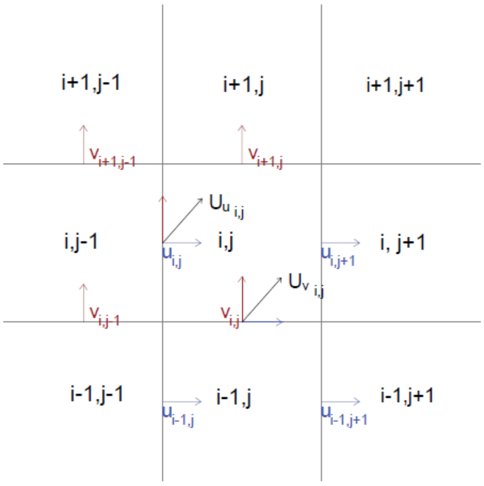

Figure 10. Option 2 for computing the velocity modulus in the cell. Here the velocity modulus is different. When the model computes the horizontal velocity component (u) the modulus is calculated in the same point where the u component is computed. The same happens with the v component. Own source.

The option one have the facility that the value calculated in the centre don’t oscillates so much between the computing of the velocities and forces in each direction. The problem is that you are taking into account the velocity downstream the turbine, so the thrust force have already been taken into account. In small grid size this effect is emphasised, provoking lower power’s output. The option two is different. The velocity taken into account to compute the thrust force and power is upstream. The problem is that with big cell sizes and depending of the direction of the flow the value oscillates more. In this implementation the first option have been chosen for the calculating of the velocity modulus even though it gives a more conservative results.

2.2.4. Output data

To sum up, the implementation is a 2D-3D model for axial turbines (with pitch blade control, able to turbine in both directions) and not discretized in the horizontal domain. To visualise the results of the simulations there is the basic output data given by MOHID Studio, like velocity components or water lever. Nevertheless some other parameter where interesting to visualize so they have been programed. They are written in the same way as the Time Series Files (these files are results of parameters that the user wants to track during the simulation and they are plotted in a x-y graph). In the appendix A.1 is explained the input data required for this implementation, where it has to be written the specific keyword in order to print the output data. The parameters plotted will be three, power of the turbine, energy extracted and velocity of the flow in the turbine cell. This three parameter are plotted for each turbine selected and for the total. In other words, you can plot the power and energy extraction of the array of turbines that you are simulating and of single turbines also, simultaneously.

3. RESULTS

The results shown in this chapter have been carried out to show the potential of the implementation, but they also serve for verifying purposes. All the results are already in steady conditions.

Simulation	Time step [s]	2D/3D	Vertical discretisation	Grid size [m]	Nº turbines	Input velocity [m/s]	Cut-in speed [m/s]	Design speed [m/s]
Horizontal channel	1	3D	25 layers equidistant.	20	1	3	1	2.5
Horizontal channel 2D	1	2D	-	20	1	3	1	2.5
Diagonal channel	1	3D	25 layers equidistant.	20	1	3	1	2.5
Array layout	1	2D	-	20	14	3	1	5
Real case	20	2D	-	300	40	Tidal current	0.5	1.5

Table 1. List of simulations. Own source

The values of the time step have been chosen in order to assure the stability of the model, they are directly related with the resolution of the grid.

3.1. GENERAL SETUP

The basic axial turbine parameters will be the same for all the simulations in order to simplify and allow the comparison of the different simulations. Only some variations will be made in the cut-in speed and design velocities and in the diameter for the real case simulations. Any change of this values will be specified. The thrust and power coefficient will be the same in all the simulations.

Turbine set-up	
Diameter	20 m
Power coefficient (C_{Po})	0.40
Thrust coefficient (C_{T0})	0.85

Table 2. Basic parameters values. Own source

The C_P and C_T coefficient values adopted for the simulations are the ones suggested by Bahaj et al in his study [12].

3.2. SIMULATIONS

3.2.1. Horizontal channel

This simulation is a basic one in order to see the effect of the turbine in the horizontal and vertical planes: flow modification, water level and energy extraction. The bathymetry of the domain is constant, 40m depth, and the turbines are placed at a height of 20 meters respect the floor. The cut-in speed and design speed are 1 and 2.5 m/s respectively. The velocity imposed in the channel left side boundary is 3 m/s.

The flow field is illustrated from figure 11 to figure 13. The modification of the flow in both planes, horizontal and vertical meets the expectations of what the flow, in a macroscopic scale, should do.

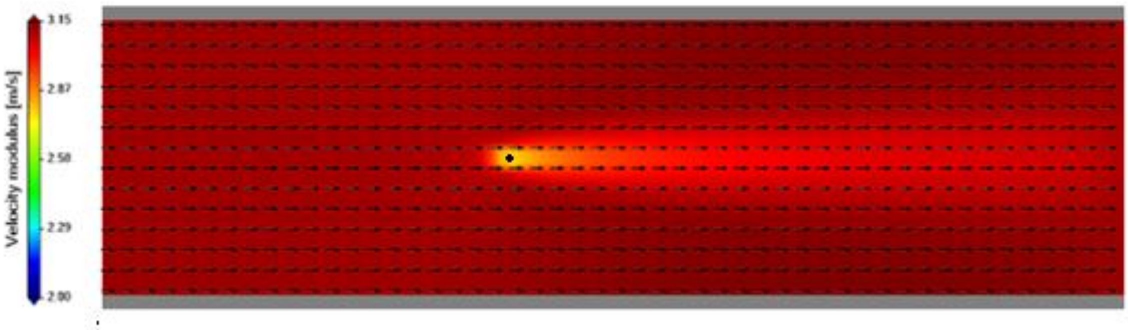

Figure 11 Velocity modulus in the horizontal plane in the layer 12, which represents a depth of 19.2 meters. Source: MOHID Studio

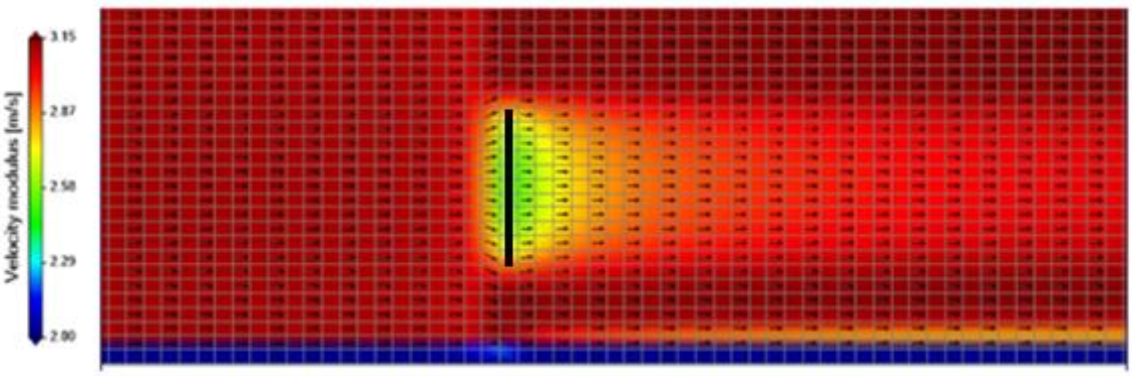

Figure 12. Vertical cut in the x axis of the flow field. In order to provide a better visualisation, a certain distortion have been applied in the horizontal dimension of the grid. As the turbine is discretized in the vertical domain, the resolution in the vertical is better than in the horizontal. Source: MOHID Studio

In figure 12 it is shown how the flow modification causes high velocities in the floor just under the turbine. As can be expected, this variation of flow velocities provoques a variation on sediment dipositation and erosion. This shows a subconsequence result of current turbines.

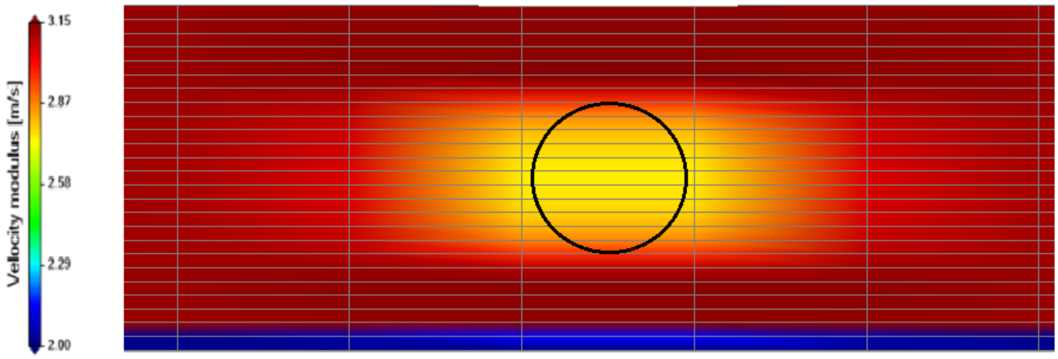

Figure 13. Vertical cut in the y axis, frontal view of the turbine. Source: MOHID Studio

In figure 13 the velocity field is inaccurate because of the shape of the grid cells in the vertical domain. In this implementation the vertical discretisation, previously explained in point 2.2.3, is thought to correct this effect in the calculations of thrust and power. The area used to calculate this parameters is the real area swept by the blades and not the vertical surface of the cells where the turbine is placed. Even though the results are not accurate because even if the value of thrust is calculated with the correct area, the force is applied in all the cell surface. To correct this distortion an horizontal discretization is needed in order to approximate the shape of the turbine to a circumference. To use complex vertical geometries for the cells in the vertical domain in order to approximate the shape to a circumference will be also an option.

The next figure shows the effect of the turbine on the water level. The effect on water level is small because the channel is too wide, yet some results can be appreciated. Upstream the turbine there is a slightly increase of the level while downstream the level decreases, as it is expected. To study better the effects on water level of current turbines in order to verify the results obtained it will be interesting to do other simulations with more than one turbine and with narrower channels.

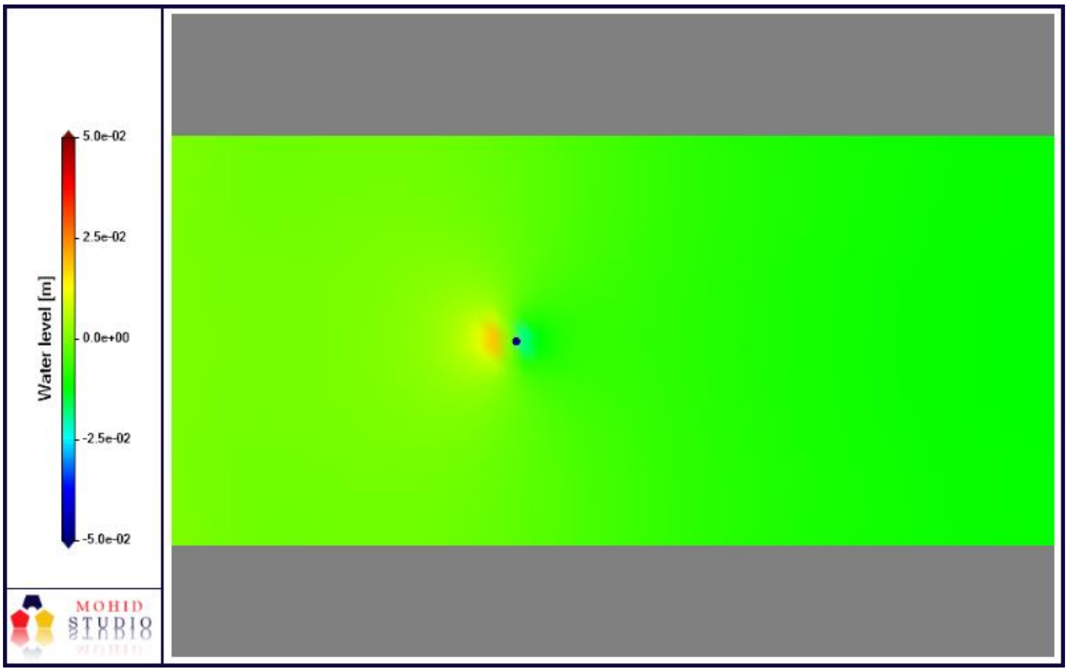

Figure 14. Effect of the turbine in the water level. This picture is made in the upper layer. Source: MOHID Studio

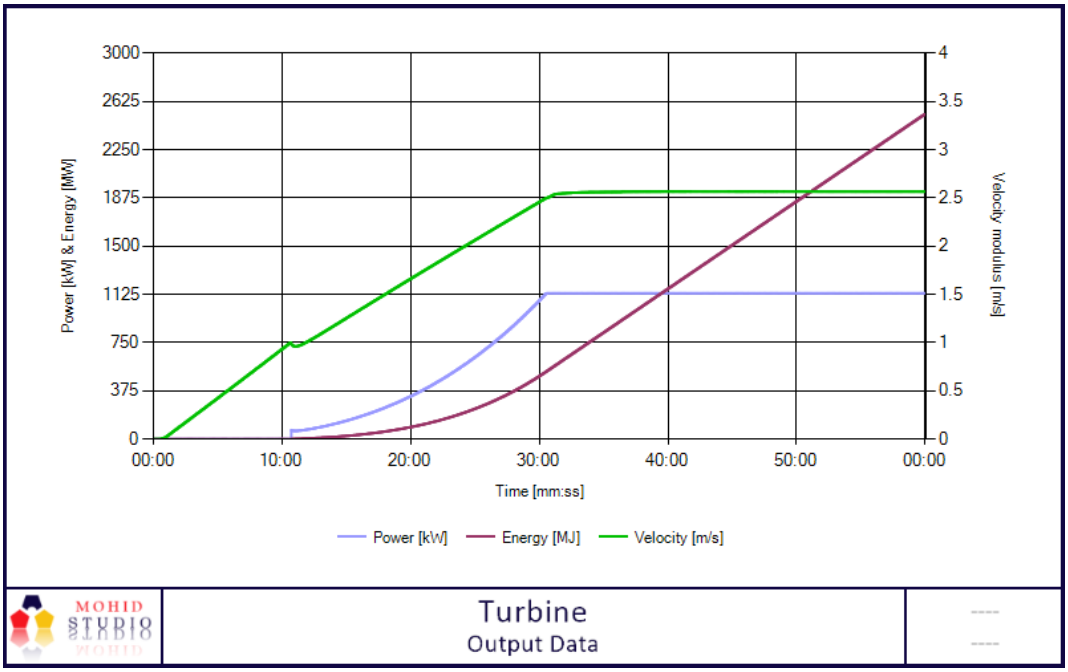

Figure 15. X-Y Graph of the velocity, power and energy output. Source: MOHID Studio

The last figure shows the power, energy and velocity outputs of the turbine. The effect of the cut-in speed and design speed can be appreciated in the power and velocity outputs. When the turbine is starting to work the velocity modulus in the cell where the turbine is placed significantly decreases and the turbine starts to produce power and energy. When the turbine reaches the rated speed, the power output becomes constant while the velocity continues to increase till its maximum.

3.2.2. Horizontal channel 2D

This simulation is the same as the previous one, the only variation is that instead of 3D is a 2D simulation. The purpose is to show the similarities and differences in the results between 2D and 3D simulations.

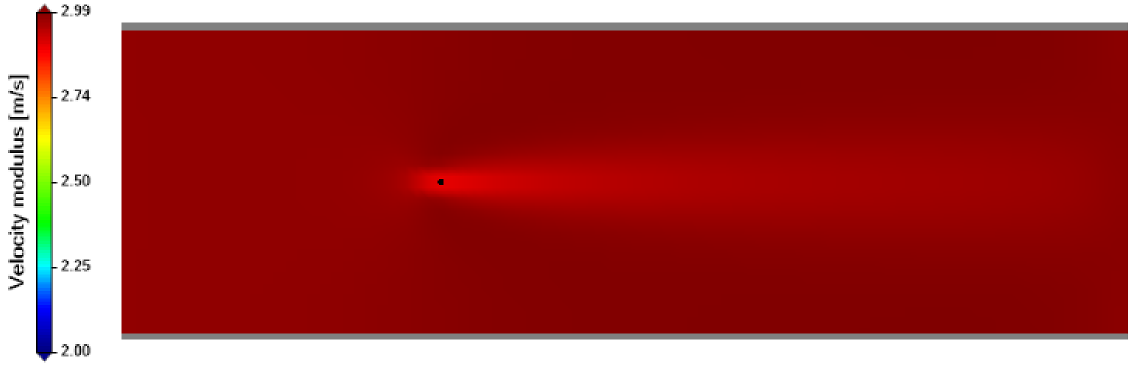

Figure 16. Velocity modulus in the horizontal plane. Source: MOHID Studio

Visually, as it is shown in figure 16, the effect of the flow is smaller in the 2D simulation than in the 3D. This is because we are working with the average velocity of all the water column in each cell. In figure 11, the results shown are from the layer 12. Here there isn't vertical discretization and the values of thrust force and velocity are for all the water column. A 2D cell in this case can be understood as a 3D cell with the bathymetry value as the vertical longitude of the cell. So, indeed, the flow of water going through the cell is higher in this simulation while the thrust force is almost the same in both simulations. That's why visually, the effects are smaller. The results in the 2D simulation, as we work with the average velocity of the cell, are directly affected by the bathymetry. Using the same turbines and same size of the cells in the horizontal domain, higher bathymetry values will reduce the effects of the turbine. That's why in shallow waters it is correct to work with 2D simulations while in deep waters a 3D discretisation is needed in order to see the effect of turbines or other devices. The difference between the area of the turbine and the area of the cell, considering the 2D cells as 3D cells, when there is no discretisation, is the main parameter to take into account in the interpretation of the results. When the size of the cell is more similar to the size of the turbine the results are more accurate.

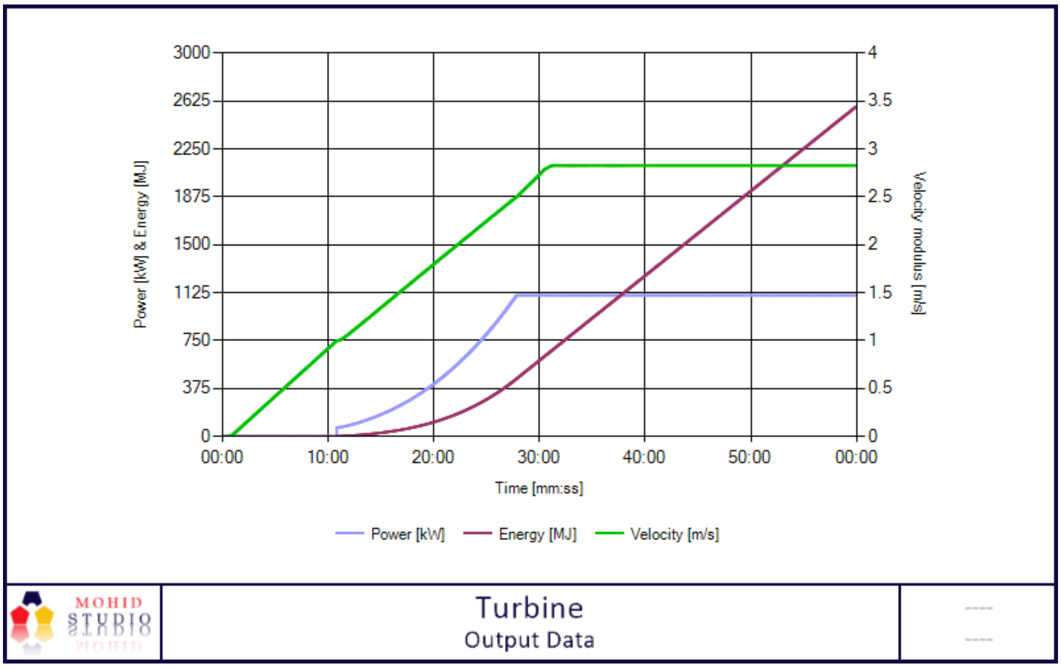

Figure 17. X-Y Graph of the velocity, power and energy output. Source: MOHID Studio

In figure 16 the output results are shown. Comparing them to the ones obtained in the previous simulation, they are pretty much the same. The main difference is in the velocity output. As we are working with the same velocity in all the water column of the cell, the effects of the turbine are less and so the maximum velocity reached is higher. The power and energy results are almost the same as in both cases the design speed of the turbine, 2.5 m/s, is reached. If not, there will be more difference in the outputs of power and energy as they depend on the velocity.

In conclusion, the results of 2D simulations in front of 3D simulations, in some cases, as the one shown in this point, are quite acceptable while we are working on shallow waters.

3.2.3. Diagonal channel

This simulation is the same as the one in the point 3.2.1. with a diagonal channel instead of a horizontal one. The idea is to prove the robustness of the implementation and that it can give good results no matter the direction of the stream, and if there are differences to analyse them. The bathymetry is the same, 40 meters depth, and in this case the cut-in speed is the same as the one before (1 m/s) and 2.5 m/s for the design speed. The velocity imposed to the left open boundary of the channel is 3 m/s and in the same direction of the channel.

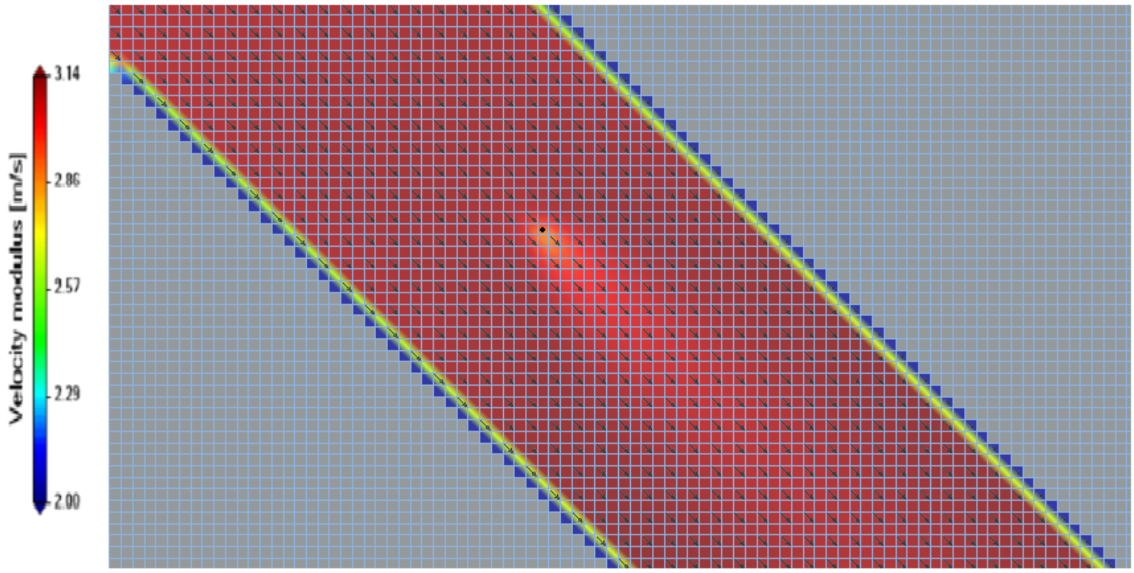

Figure 18. Velocity field in the layer 12 (19.2 metres depth). Source: MOHID Studio

Comparing the velocity field obtained in the diagonal channel (figure 18) with one obtained in the horizontal one (figure 11) there is a difference in the wake shape. The velocity going through the turbine is bigger in this case and consequently the wake is smaller. This means that the thrust force is different in both simulations, while it should be the same. So there is some kind of relation between the relative direction of the flow and the cell where the turbine is placed. In order to see properly the difference between the velocities in both cases, the results of the simulations are plotted in the next figure.

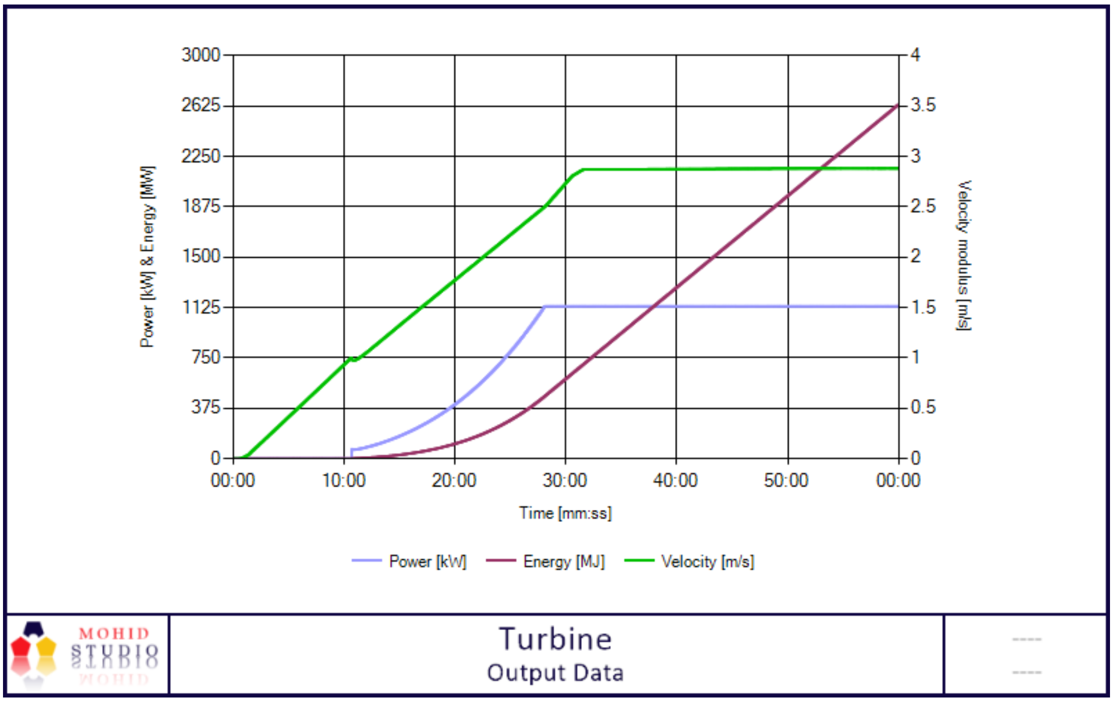

Figure 19. X-Y Graph of the velocity, power and energy output. Source: MOHID Studio

While the final power output is the same, the energy and the velocity values are different. To compare both simulations we will focus on the velocity value as the other parameters are velocity dependants. The velocity modulus is higher in this case in a value around 0.2-0.3 m/s. At first sight it seems that the difference is caused because in the diagonal case, as it is shown in figure 18, the velocity modulus is not perpendicular to the velocity direction as it happens to be in the simulation shown in point 3.2.1.It seems to be related with the effect explained in the 2D simulation (point 3.2.1) where the quantity of water going through the cell affects the results. But in this case the increase of water flow going through the cell is caused by the geometry of the cell and the direction of the flow respect the cell surfaces. In this case we have two components of the velocity u and v, and they are computed in the centre of west and south faces respectively. Also the cell geometry used is quadratic. So, considering that in each cell there are two flows of water going through it, one due to the u component and the other due to the v component, the total amount of water going through the cell is higher. This is what seems to cause this effect of higher value on the velocity going through the turbine.

It will be necessary to do further studies on this issue in order to verify it, but it seems that with a properly horizontal discretisation or a proper cell geometry the results will be more similar compared to the horizontal channel. In this case for example, reducing the cell size so the diagonal of the cell is the same as one face of the quadratic cell used, the results should be more similar.

3.2.4. Array layout

Here the results of two simulations are shown with the intention of exposing the capability of the implementation for array layout studies. Two different scenarios are contemplated, the main characteristics are the same, the only difference between them is the distribution of the turbines in the domain.

The domain is a channel of 2 km long and 540 m width, same grid size in both cases and a constant bathymetry of 40m depth. The velocity of the current stream is imposed as a boundary condition, 3 m/s. In this study, the design velocity for the turbines has been modified to 5 m/s (the value itself doesn’t matter, it has to be higher than the stream velocity), so that the turbines never reach their design speed. This is done in order to see the difference between both arrays with the power and energy outputs. The number of turbines in both scenarios is the same, 14.

The first layout is a three-lined array (5 - 4 - 5). The y-axis distance between turbines is 20 m while the x-axis distance is around 80m. The second layout is a four-lined array (4-3-4-3) divided in two main lines with an x-axis distance of 160 meters. Both distributions are shown in the figures 21 and 22 respectively.

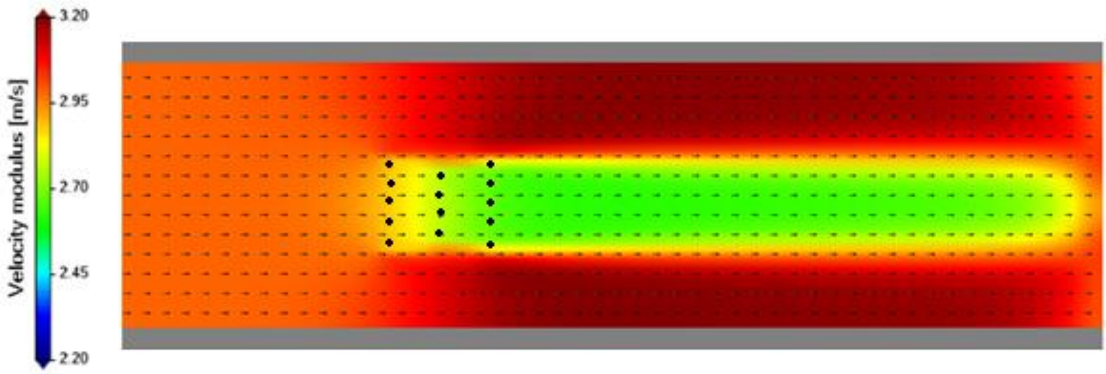

Figure 21. Velocity field for the layout 1. Source: MOHID Studio.

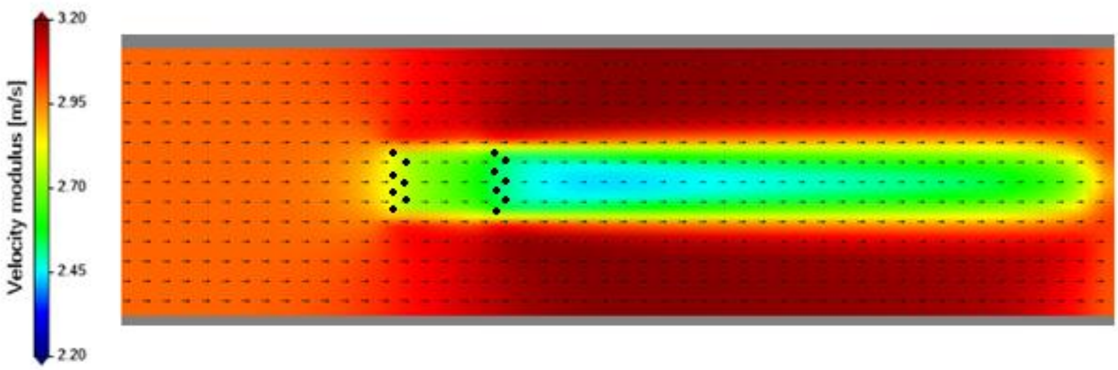

Figure 20. Velocity field for the layout 2. Source: MOHID Studio

Taking a look at the flow velocities field of the layout 2, we can see how the second layout creates lower velocities after the two last lines of turbines. This is because the turbines aren't as separated in the y direction as in the layout 1, creating a blockage effect. Another thing to highlight is the shape of the wakes at the end of the channel in both distributions. They end very fast which is a little unrealistic. It seems that the longitude of the channel affects the shape of the wake. Further studies with longer and shorter channels should be carried out in order to understand why this happens. Below the graphs of the power output and velocity of both arrays are shown.

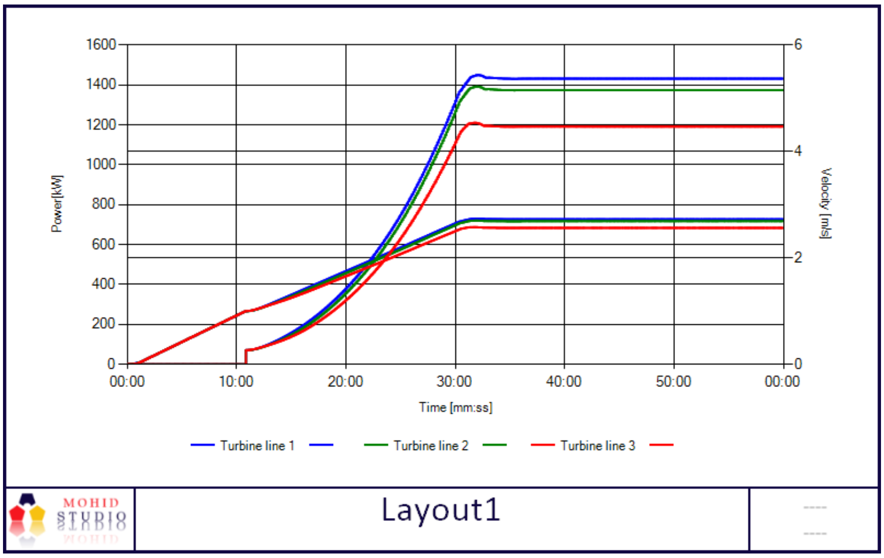

Figure 22. Power and velocity data for the layout 2. The values shown are from three turbines, one of each line of the array. Source: MOHID Studio

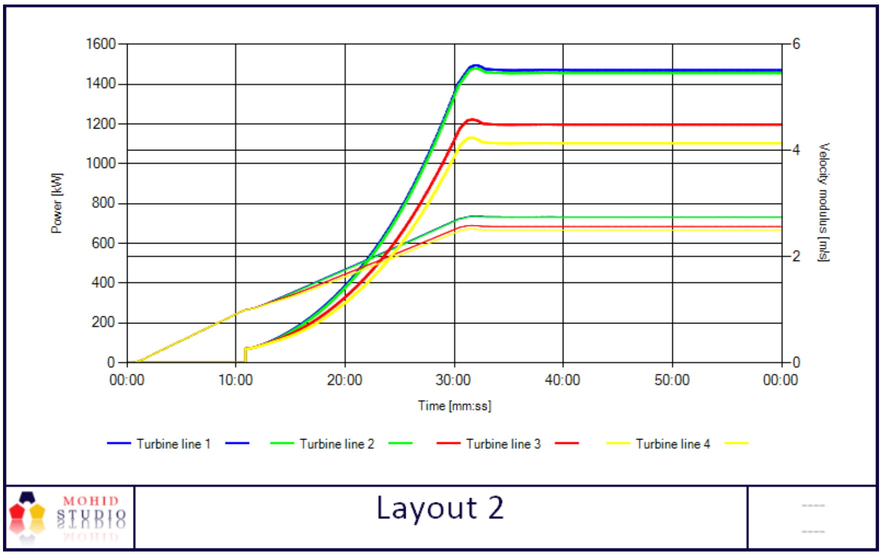

Figure 23. Power and velocity data for the layout 1. The values shown are from three turbines, one of each line of the array. Source: MOHID Studio

Both graphics show the effect of the layout for each distribution in the velocity and power output. Each graph shows the values of velocity and power of one turbine of each line of the array. These graphs allow the user to see how the position of the turbines influence their performance. In other words, they represent the interaction between the turbine lines. For example, in the fig 23, you can notice how the effect of the line one respect the line two is almost inappreciable while the line 3 respect the line 4 is greater.

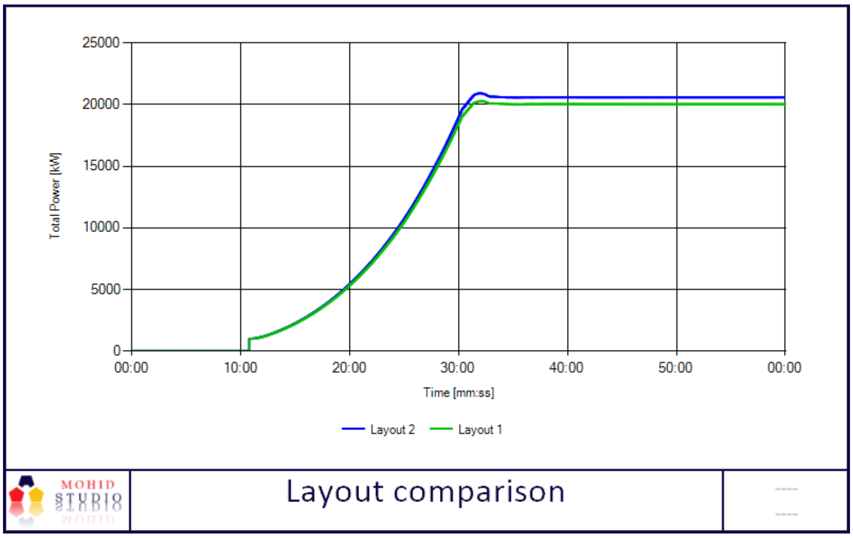

Figure 24. Total power extraction of both arrays. Numerically the layout 2 reach a maximum power output of 20.59 MW while the layout 1 reach 20.03 MW.

Finally, to compare both layouts, the total power is plotted. The fig 24 shows that the second layout is better in terms of energy extraction. This is just an example, multitude of layouts can be studied and compared between them. It also allows a more detailed study of how the relative position of the turbines affects the flow and their performance.

3.2.5. Real case

In this case the implementation is tested with a tidal simulation to see the results of energy extraction in a “real” scenario. Nevertheless, the visual data available for flow modification is not accurate because of the low resolution of the simulation (the size of the cell is quite big compared to the other simulations). So, as the implementation allows to put more than one turbine per cell, this simulation will emulate a turbine farm. Instead of defining an array of turbines, a density of turbine per cell will be defined. In this case, considering that the cells where we will place the turbines have the approximate size of 300x300m, we will place in 4 cells a total of 40 turbines, 10 turbines in each cell.

The emplacement chosen to make the simulation is the Tagus Estuary (fig 25). Is not an idyllic emplacement to install tidal turbines because the current streams are weak and the bathymetry of the estuary is not too depth. But for the purpose of testing the implementation in low resolution grids it will be enough. The emplacement chosen has enough depth to handle little turbines of 10 meter diameter. The cut-in speed has been changed to 0.5 m/s and the design velocity to 1.0 m/s.

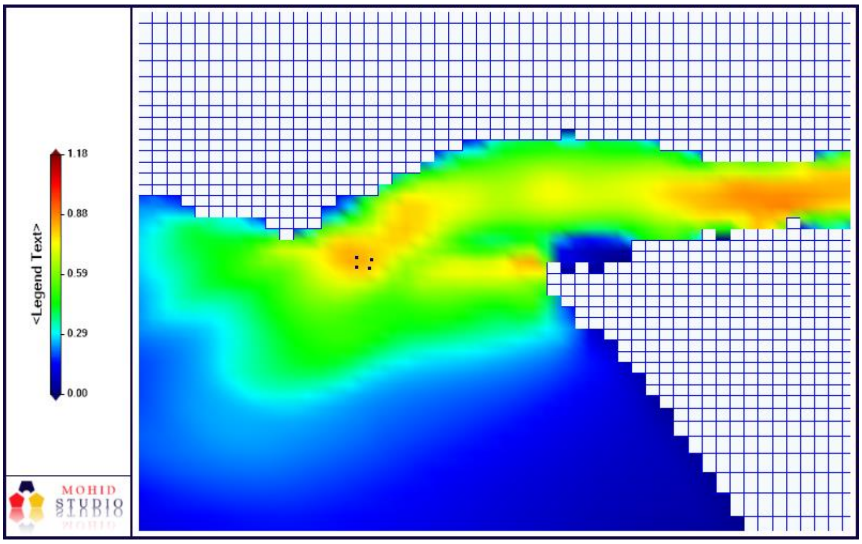

Figure 25. Placement of the turbines in the Tagus Estuary. Each point represents one group of 10 turbines, and each group is placed in a cell of 300x300m. Source: MOHID Studio.

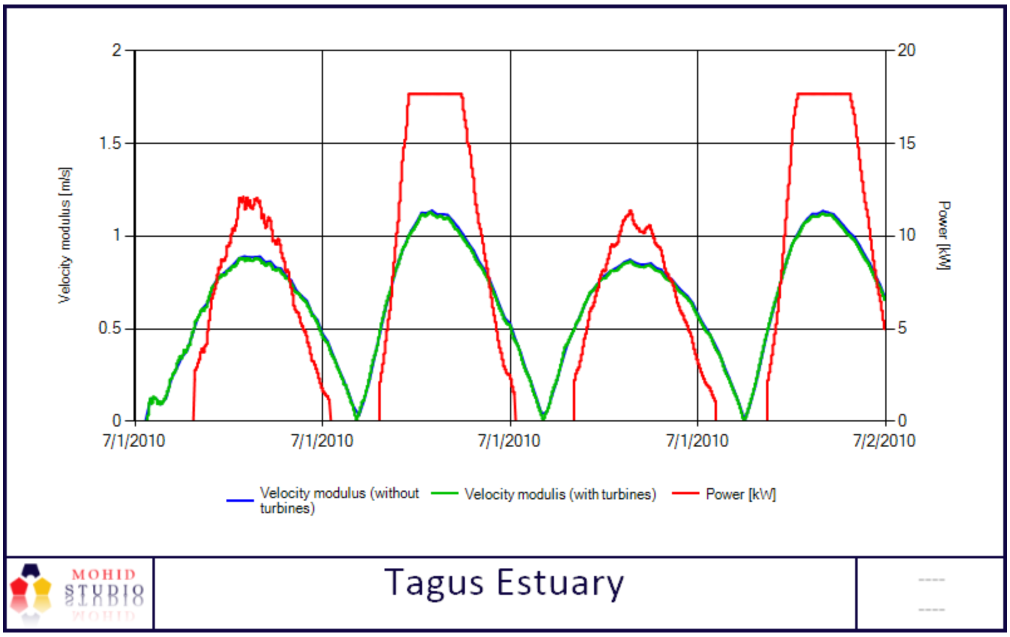

Figure 26. Power and velocity output of one group of 10 turbines. Source: MOHID Studio

The velocity in both cases, with and without turbines, is calculated in the centre of the cell. The difference is so small because the cell is big, 300x300 meters, the density of turbines per cell is small (10 turbines per cell), the turbines are small, 10 meters diameter, and the velocity range where they can work is small. Basically, the power output is small and that's why the effect of the turbines in the flow velocity is so small. Finally the values of total power and energy output are plotted.

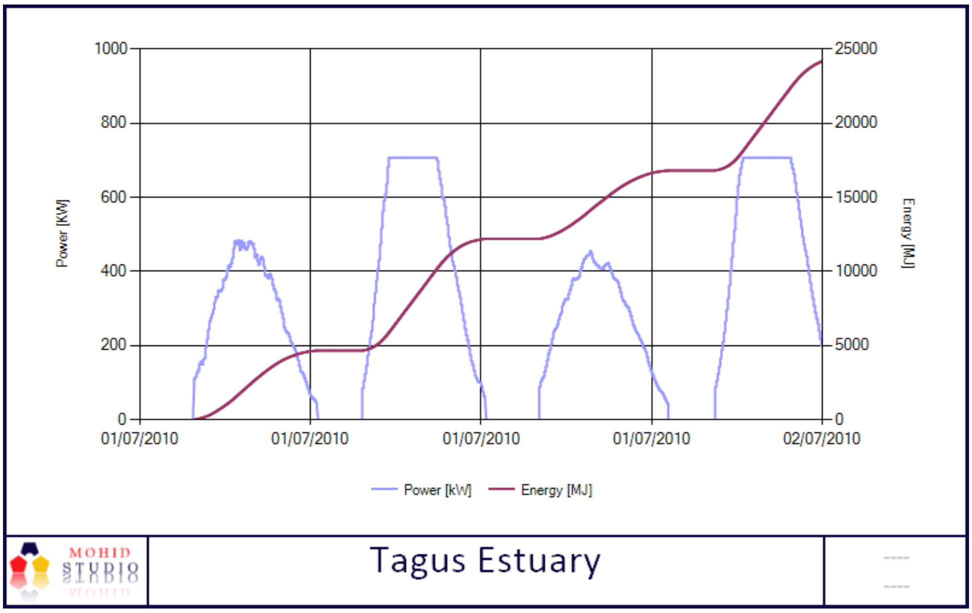

Figure 27. Energy and power output of the 40 turbines placed in the Tagus Estuary. Source: MOHID Studio

4. CONCLUSION

4.1. ACHIEVEMENTS

A good compromise between the computational cost and the results has been achieved. The implementation capabilities are quite good and they allow some interesting studies of energy extraction, array layout and flow modification.

The implementation is a reliable and simple model of the effect of turbines inside a realistic and complex three-dimensional hydrodynamic model. Despite the verification of the implementation is not complete, the objectives proposed have been achieved in a proper way, and some good ideas have appeared due to the realisation of this project.

4.2. LIMITATIONS

The origin of the main limitations of this implementation is the non-discretisation in the horizontal plane.

The implementation is not programed to work with really small grid size, the results will be always inappropriate when the cell size is under the turbine diameter thought there is no discretisation. Also it is not prepared to work with really large grid size, because visual data doesn’t show significant changes due to the resolution and also the calculation are not really accurate.

Another limitation derived of the horizontal discretisation is the layout array studies. Actually there are some limitations with the distances and positions between the turbines because the minimal distance between them will be, at least, as big as the grid cell size diameter. These are the main limitations of the implementation.

4.3. FUTURE WORK

4.3.1. Verification of the implementation

The main point of future work is to validate the model and the results obtained with real data or other models available that are already verified. While the implementation seems robust and the results, taking into account the limitations of the model, seem realistic, further analysis need to be done. It will be interesting to compare the results of a single turbine with a CFD simulation in order to see the differences of both simulations.

After a first step for validating the results with other simulation tools, the implementation is ready to be tested in a real environment, also with verifying purposes. In this case, it will be of great interest to squeeze the potential of MOHID and work with nested models. Nested models are the perfect tool for solving the resolution problem arising from the difference in scale of the different processes involved in tidal energy: tides and turbines. The idea is to have a high resolution where the turbines are placed, and work with lower resolution in the rest of the domain. In the actual implementation the grid size of the nested model should be as small as the turbine rotor diameter.

Once the verification process is completed and the results are satisfactory, it will be interesting to test the implementation with sediment transportation to see the influence of turbines in this process. It is a study of great interest in order to contribute with some data on the environmental impact of this kind of technology.

4.3.2. Improvement of the implementation

Besides testing the implementation, there are some improvements that can be done. The more affordable one is to improve the input data format in order to make it more accessible and efficient to create it. Actually, if the user needs to work with large arrays of turbines, he will spend a good time configuring it. In the appendix A.1 the actual format of input data is explained. The easiest way to do it is to link the input data of the turbines with a xyz format file with only the location of the turbines. This is interesting because MOHID actually provides a tool to place points in the map and saves them in this kind of files. The model will read the basic and common parameters (thrust and power coefficients, dimensions, etc) in one file, and the locations in another file. On one hand, this restricts the model in the sense that all the turbines will have the same basic parameters, but in the other hand, it makes the creation of input data more efficient.

The second improvement is to implement the horizontal discretisation with the purpose of overcoming the limitations exposed in the point 4.2. The following approach has not been programed because there were some complications in the transformation of the input data to the discretized geometry, which is considered the main issue. The idea of this implementation is to use a similar input data file and the implementation is the one in charge of creating the geometry of the turbine in the domain, either in 2D or 3D simulations.

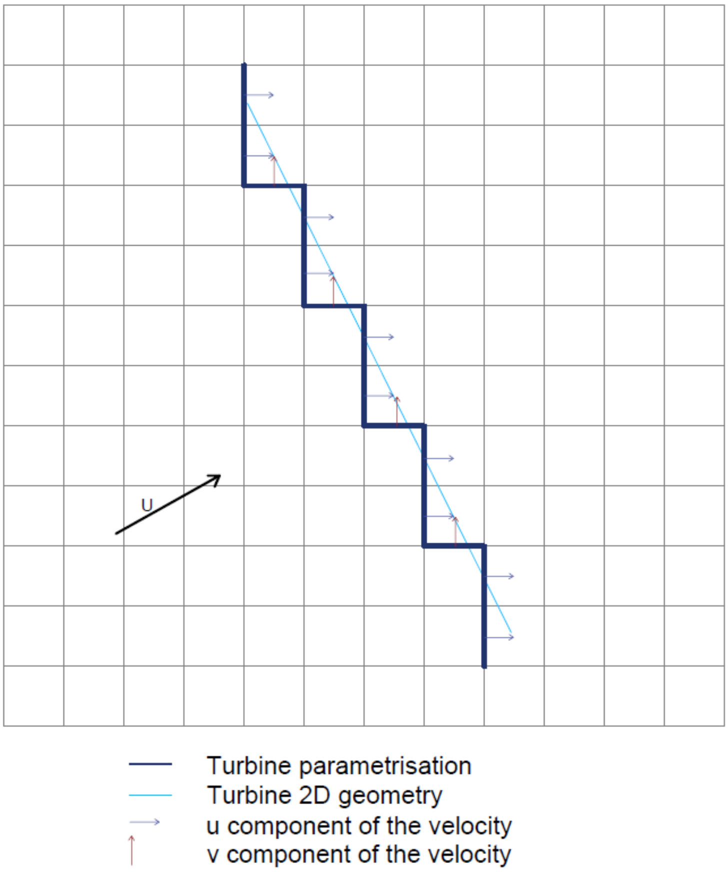

Figure 28.Example of 2D discretization. Own source.

Figure 28 shows a possible discretisation of the turbine in the 2D domain. This discretisation only computes the u and v components of the velocity in some cells, in order not to duplicate the effect. To make so this discretisation has only the location and the diameter of the turbine, we need to draw the parameterisation perpendicular with the velocity direction, and then make the discretisation in the domain. Here the calculation of the modulus of the velocity for computing the force should be calculated as the option 2 presented in the point 2.2.3. For the power and energy outputs the velocity modulus can be an average as it’s done in the vertical direction in the actual implementation.

Also the direction of the turbine should be corrected in case that the velocity direction changes because in this model the perpendicularity is not assured. The direction of the turbine can be obtained with the sum of the longitude of the faces where the u component of the velocity is computed and the longitudes of the cells where the v component is computed. With this we can check if the perpendicularity between velocity and turbine is conserved during the simulation and correct the geometry in case that the deviation is significant.

For the 3D model, the discretisation will be in both vertical and horizontal domains, so the drawing of the turbine will be similar to a circumference. The actual vertical discretisation will be useless in this model, so a new one should be made, applying the force to the surface of cells that are in a distance from the turbine position lower than the radius. Nevertheless, once the horizontal discretisation is made, the vertical should be more easy because the i and j values of the cells affected are the same as in the ones discretised in the horizontal domain, only the k dimension should be determined with the rule of the distance to the centre of the turbine.

The future implementation presented in this point is expected to work properly with high resolution grids in a range of five to ten times less the turbine diameter. It has to be taken into account that this implementation, if it wants to be used in tidal simulations with real environments, could have a very high computational cost. Variable grids or nested domains can decrease the heaviness of the calculations. With the implementation described, a good resolution to take advantage of it will be in the scale of 1 meter while normally, in hydrodynamic coastal studies the grid size is not lower of 10 meters.

There is still work to do to improve the implementation, and much more work to do to achieve the goal of a sustainable future for the energy system. This project, in the humblest way possible, tries to become a tool implemented in the MOHID Studio application in order to boost current energy extraction, embracing the philosophy of MOHID as the decision support tool that it is today.

5. REFERENCES

[1] NASA. (n.d.). https://climate.nasa.gov/. Retrieved from: https://climate.nasa.gov/vital-signs/global-temperature/

[2] International Energy Agency. (2017). World energy balances: Overview. Retrieved from: http://www.iea.org/publications/freepublications/publication/WorldEnergyB WorldEn2017Overview.pdf

[3] International Energy Agency. (2017). Electricity information: Overview. Retrieved from: http://www.iea.org/publications/freepublications/publication/ElectricityInforE lectr2017Overview.pdf

[4] International Energy Agency. (2017). Renewables information: Overview. Retrieved from: http://www.iea.org/publications/freepublications/publication/RenewablesInf Renewabl2017Overview.pdf

[5] Brito E Melo A, Village JL, editors. Annual report 2016. Implementing Agreement on ocean energy systems. IEA-OES

[6] Andreas Uihlein, Davude Magagna (2015).
Wave and tidal current energy – A review of the current state of research beyond technology

[7] John M. Bane, Ruoying He, Michael Muglia, Caroline F. Lowcher, Yanlin Gong, Sara M. Haines. (2017).
Marine Hydrokinetic Energy from Western Boundary Currents. *The Annual Review of Marine Science*.

[8] MARETEC research department. MOHID Modelling System Description. Retrived from: http://www.maretec.mohid.com/portugueseestuaries/Reports/MOHID_Des Descript.pdf

[9] Ramiro Neves. Instituo Superior Técnico.
Numerical models as decision support tools in coastal areas.

[10] Arakawa, A. and V.R. Lamb (1977) - Computational design of the basic dynamical processes of the UCLA General Circulation Model. Methods of Computational Physics, 17, pp.174-264

[11] David R. Plew, Craig L. Stevens
Numerical modelling of the effect of turbines on currents in a tidal channel – Tory Channel, New Zeland (2013)

[12] a. S. Bahaj, a. F. Molland, J. R. Chaplin, and W. M. J. Batten, “Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank,” Renewable Energy, vol. 32, no. 3.

[13] https://github.com/Mohid-Water-Modelling-System/Mohid

APPENDIX A

A.1. INPUT DATA

The input data file is where the list of the turbines and their parameters have to be specified. The data is organized with the following keywords:

Global keywords		
Keyword	Default value	Description
TIMESERIE	0	This is the global parameter of the output files, if you want to write any output of any of the turbines this parameter should be 1, if not, 0.
<beginturbinelist>	-	Indicates the begin of the turbine list.
<endturbinelist>	-	Indicates the end of the turbine list
<<beginturbine>>	-	Indicates the start of the parameters of a single turbine
<<endturbine>>	-	Indicates the of the parameters of a single turbine
Specific keywords		
DIAMETER	-	The diameter of the turbine, in meters
HEIGHT	-	The heigh of the centre of the turbine respect the floor, in meters
CP	-	Power coefficient value
CT	-	Thrust coefficient value
LOWER_VEL	0	Cut-in speed, in m/s
UPPER_VEL	10	Design speed, in m/s
POS_LONG	-	Longitude position in geographic coordinates and x position in metric coordinates.
POS_LAT	-	Latitude position in geographic coordinates or y position in metric coordinates.
TIMESERIE	0	1: activates the timeserie module and prints the output data of the turbine 0 : no output data

Table 3. Input Data keywords. Own source

An example of input data will be:

```
TIMESERIE          : 1

<beginturbinelist>
<<beginturbine>>
DIAMETER           : 20
HEIGHT             : 20
CP                 : 0.45
CT                 : 0.85
LOWER_VEL          : 1
UPPER_VEL          : 2
POS_LONG           : 789.70291822
POS_LAT            : 988.39745136
TIMESERIE          : 1
<<endturbine>>

<<beginturbine>>
DIAMETER           : 20
HEIGHT             : 20
CP                 : 0.45
CT                 : 0.85
LOWER_VEL          : 1
UPPER_VEL          : 2
POS_LONG           : 729.70291822
POS_LAT            : 988.39745136
TIMESERIE          : 1
<<endturbine>>
<endturbinelist>
```

Figure 29. Example of input data. Own source

A.2. MOHID CONFIGURATION FILES

In order to activate the implementation, some modifications need to be done to the main configuration files. Two files need to be changed, the hydrodynamic.dat file and the nomfich.dat.

In the hydrodynamic file the keyword TURBINE needs to be written with a value of 1 for activating the implementation. If it is not written or with a value of 0 it will not work.

Hydrodynamic_1.dat

```
!ADV_METHOD_V                   : 1
!TVD_LIMIT_H                    : 4
!TVD_LIMIT_V                    : 4

!VOLUME_RELATION_MAX          : 1.3

CORIOLIS                      : 0

NONHYDROSTATIC                : 0

TIDE                          : 0
WIND                          : 0
WATER_DISCHARGES              : 0
MOMENTUM_DISCHARGE            : 0
BAROCLINIC                    : 0

TURBINE                       : 1

DATA_ASSIMILATION             : 1

RADIATION                     : 2
```

Figure 30. Example of hydrodynamic file with the implementation activated. Own source

The file Nomfich.dat is where the routes of the files needed for the simulations are specified. So, in order to make the implementation work and that the model can read the input data of the turbines, this route should be written in this file. The keyword for this file is TURBINE.

```
IN_BATIM          : D:\MOHID Water Quick Start Guide\Projects\11Set\General Data\Digital Terrain\gridData.dat
ROOT              : D:\MOHID Water Quick Start Guide\Projects\11Set\res
ROOT_SRT          : D:\MOHID Water Quick Start Guide\Projects\11Set\res\Run1\
SURF_DAT          : D:\MOHID Water Quick Start Guide\Projects\11Set\data\Atmosphere_1.dat
SURF_HDF          : D:\MOHID Water Quick Start Guide\Projects\11Set\res\Atmosphere_1.hdf
DOMAIN            : D:\MOHID Water Quick Start Guide\Projects\11Set\data\Geometry_1.dat
IN_DAD3D          : D:\MOHID Water Quick Start Guide\Projects\11Set\data\Hydrodynamic_1.dat
OUT_DESF          : D:\MOHID Water Quick Start Guide\Projects\11Set\res\Hydrodynamic_1.hdf
OUT_FIN           : D:\MOHID Water Quick Start Guide\Projects\11Set\res\Hydrodynamic_1.fin
BOT_DAT           : D:\MOHID Water Quick Start Guide\Projects\11Set\data\InterfaceSedimentWater_1.dat
BOT_HDF           : D:\MOHID Water Quick Start Guide\Projects\11Set\res\InterfaceSedimentWater_1.hdf
BOT_FIN           : D:\MOHID Water Quick Start Guide\Projects\11Set\res\InterfaceSedimentWater_1.fin
AIRW_DAT          : D:\MOHID Water Quick Start Guide\Projects\11Set\data\InterfaceWaterAir_1.dat
AIRW_HDF          : D:\MOHID Water Quick Start Guide\Projects\11Set\res\InterfaceWaterAir_1.hdf
AIRW_FIN          : D:\MOHID Water Quick Start Guide\Projects\11Set\res\InterfaceWaterAir_1.fin
IN_MODEL          : D:\MOHID Water Quick Start Guide\Projects\11Set\data\Model_1.dat
IN_TIDES          : D:\MOHID Water Quick Start Guide\Projects\11Set\data\Tide_1.dat
IN_TURB           : D:\MOHID Water Quick Start Guide\Projects\11Set\data\Turbulence_1.dat
TURB_HDF          : D:\MOHID Water Quick Start Guide\Projects\11Set\res\Turbulence_1.hdf
DISPQUAL          : D:\MOHID Water Quick Start Guide\Projects\11Set\data\WaterProperties_1.dat
EUL_HDF           : D:\MOHID Water Quick Start Guide\Projects\11Set\res\WaterProperties_1.hdf
EUL_FIN           : D:\MOHID Water Quick Start Guide\Projects\11Set\res\WaterProperties_1.fin
ASSIMILA_DAT      : D:\MOHID Water Quick Start Guide\Projects\11Set\data\Assimilation_1.dat
ASSIMILA_HDF      : D:\MOHID Water Quick Start Guide\Projects\11Set\res\Assimilation_1.hdf
TURBINE           : D:\MOHID Water Quick Start Guide\Projects\11Set\data\TurbineParameters_1T.dat
```

Figure 31. Example of Nomfich.dat file with the turbine input data path included at the end. Own source