# Extended Abstract

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

**Óscar Balsells Badia**

Instituto Superior Técnico, Portugal

September 2017

**Abstract** – The present document describes the project of including into the hydrodynamic model of MOHID Studio application the effect of current energy extraction with turbines. It allows users to study flow modification, energy extraction and other sub-consequent effects of the turbines. The implementation is thought for axial turbines that can turbine on both direction, 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 discretised in the vertical direction (for 3D simulations only), not in the horizontal plane.

*Keywords* – Tidal energy, Current Energy, MOHID Water, MOHID Studio, Turbine, Finite Elements

## 1. Introduction

Human activity, mainly the fact of burning fossil fuels for obtaining energy, is affecting in a non-sustainable way the environment: CO2 pollution is reaching record values, the mean global temperature has increased in almost 1 ºC since 1880 [1], the urban air-pollutions is reaching unhealthy levels, etc. 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.

Renewable energies are the cornerstone for a sustainable future, and due to the increasing demand of the electricity consumption in the years to come, the electricity industry is one of the main points to take action on. At the present time, only the 22-24% (figure 1) of the electricity comes from renewable sources, so there is still work to do in order to arrive to more reasonable numbers.

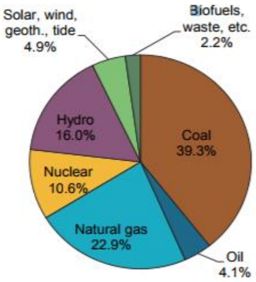

*Figure 1. World gross electricity production (%) by source in the year 2015. Source: IEA [2]*

There are many kinds of renewable energies: biomass, eolic, geothermic, marine, tidal, solar, etc. Several of this renewable sources present the challenge of being weather dependant, 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.

Of all the renewable sources this thesis is focused in current energy. There exist, nowadays, three different origins of this kind of energy: marine currents, tidal currents and river currents. All of them use the same technology to extract energy, turbines. And the energy extracted comes from the same source, the kinetic power of the water:

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

Even though current streams are a renewable sources of energy, as all of the renewable energy technologies, they aren't environmentally friendly by definition [3]. All the activities involved in the manufacturing and maintenance have an impact on the environment. The alteration in sediment transportation and the impact on the fauna habitat are some of the main effects of this technology. Further studies need to be done in this area.

The implementation done in this thesis is programmed in MOHID Modelling System, 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). All the implementation is programmed with FORTRAN 95 with an object oriented philosophy, the same as in MOHID.

## 2. Hydrodynamic fundamentals

The hydrodynamic model of MOHID solves the three-dimensional incompressible primitive equations assuming hydrostatic equilibrium, Boussinesq and Reynolds' approximations [8]. All the equations here are written in the differential form and 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, with incompressible fluid (constant density), is:

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

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, and $A_H$ and $A_V$ are the turbulent viscosities in the horizontal and vertical directions. The $\rho_0$ and $\rho$ are the reference density and density respectively, and the p is the pressure.

The discretization in time is made through a semi implicit ADI (Alternate Direction Implicit) algorithm. Basically it computes each velocity component in the horizontal, u and v, alternatively.

## 3. Description of the model

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 simulate the turbine, two main parameters need to be introduced [4]:

- The thrust force produced by the turbine rotor due to energy extraction, eq. 6.
- The power extracted by the turbine, eq. 7.

$$F_T=\frac{1}{2}\rho A_T C_T U^2 \qquad \text{Eq. 6}$$

$$P_T=\frac{1}{2}\rho A_T C_p U^3 \qquad \text{Eq. 7}$$

The $\rho$ is the density, 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 [4].

$$C_T=\begin{cases}0 & \text{sii} \quad U\le U_C\\ C_{T0} & \text{sii} \quad U_C<U\le U_D\\ C_{T0}\dfrac{U_D^3}{U^3} & \text{sii} \quad U>U_D\end{cases} \qquad \text{Eq. 8}$$

For the power coefficient, the parameterisation is the same:

$$C_P=\begin{cases}0 & \text{sii} \quad U\le U_C\\ C_{P0} & \text{sii} \quad U_C<U\le U_D\\ C_{P0}\dfrac{U_D^3}{U^3} & \text{sii} \quad U>U_D\end{cases} \qquad \text{Eq. 9}$$

Where $C_{p0}$ and $C_{T0}$ are the design values for each coefficient and $U_C$ and $U_D$ are the cut-in and design speed.

## 4. Discretization

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 is made with the equation 6. The perpendicularity between the turbine and the flow is implicitly assured by the model. In the figure 2 a visual croquis of the discretisation is shown.

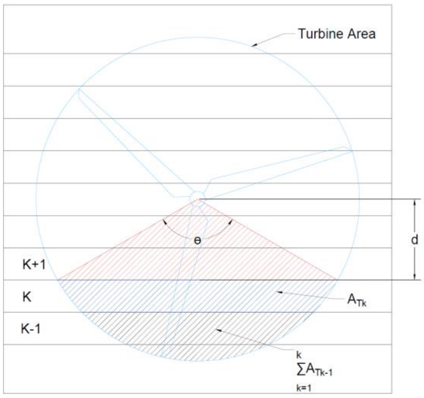

*Figure 2. Vertical discretization of the turbine area. Own source.*

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 6 and 7 can be rewritten, for a turbine placed in the cell (I,J) coordinates, as:

$$\vec{F}_{TK} = \frac{1}{2}\rho A_{T_k} C_T U_k \vec{U}_k \qquad \textit{Eq. 10}$$

$$P_T = \frac{1}{2}\rho A_T C_p U_{AV}^3 \qquad \textit{Eq. 11}$$

It is important to highlight for the spatial discretization that tehe grid in MOHID is staggered in the horizontal in an Arakawa C manner [6]. Where  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:

$$U_{AV} = \frac{\sum_K A_{T_K} * U_K}{\sum_K A_{T_K}} \qquad \textit{Eq. 11}$$

The velocity modulus is calculated in centre of the cell as an average of the consecutives cells for each direction.

## 5. Results

The input parameters for the simulations presented here have the same values for power and thrust coefficients. These values comes from the ones suggested by Bahaj et al in his study [5].

| **Turbine set-up** | |
|---|---|
| **Power coefficient ($C_{Po}$)** | 0.40 |
| **Thrust coefficient ($C_{T0}$)** | 0.85 |

### 5. A. 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 in 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. In the next figures, from 3 to 5, the velocity field and the water level can be observed.

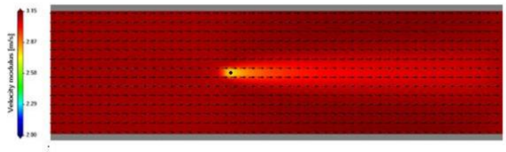

*Figure 3. Velocity modulus in the horizontal plane in the layer 12, what represents a depth of 19.2 meters. Source: MOHID Studio*

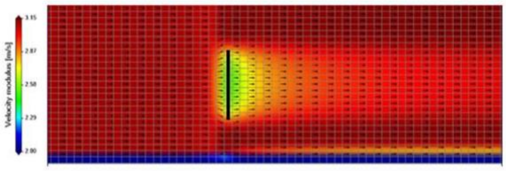

*Figure 4. Vertical cut in the x axis of the flow field. Source: MOHID Studio*

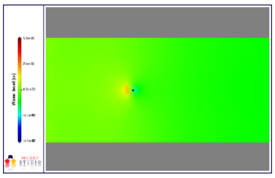

*Figure 5. Effect of the turbine in the wather level. This picture is made in the upper layer. Source: MOHID Studio*

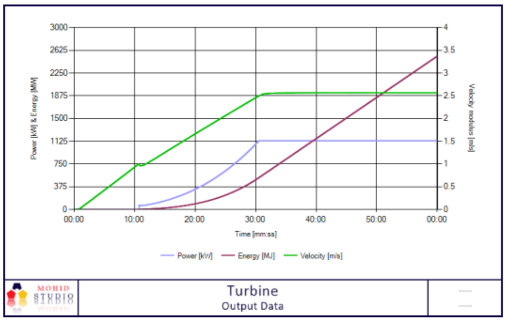

*Figure 6. X-Y Graph of the velocity, power and energy output. Source: MOHID Studio*

### 5. B. Array Layout

Two different scenarios are contemplated, the main characteristics are the same, and the only difference between them is the distribution of the turbines in the domain. The number of turbines is the same in both, 14.The domain is a channel of 2 km long and 540 m width, same grid size in both cases and constant bathymetry of 40m depth. The velocity imposed of the current stream is 3 m/s and the design speed is modified to 5 m/s. The different distribution of the turbines is shown in the figure 7.

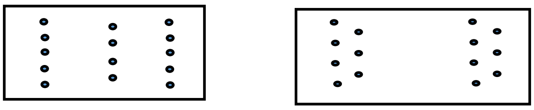

*Figure 7. Array layout distribution for option 1 and 2 respectively. Own Source.*

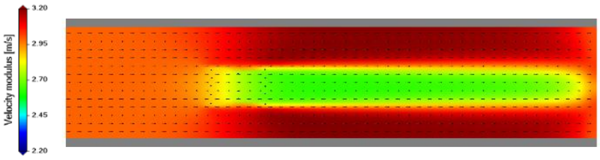

*Figure 8 Velocity field for the layout 1. Source: MOHID Studio.*

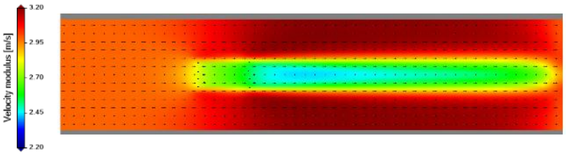

*Figure 9. Velocity field for the layout 2. Source: MOHID Studio*

Taking a look at the flow velocities, it can be appreciated 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.

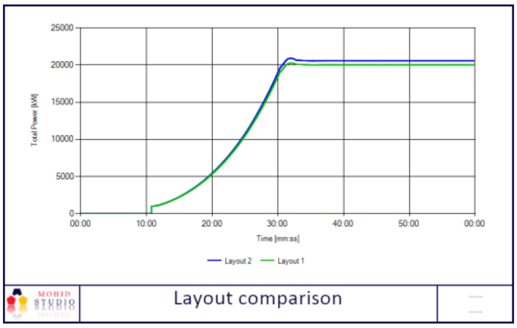

*Figure 10. Total power extraction of both arrays. Numerically the layout 2 reach a maximum power output of 20.59 MW while the layout 1 reach 20.03 MW.*

## 5. C. "Real Case"

Just in order to see the output of velocities and power in a case where the velocities come from tides and with an inappropriate scale of the grid (big grid size compared with the turbines diameter), a study of putting 40 turbines in groups of 10 turbines per cell have been done in the Tagus Stuary. The outputs are shown in the figure 11.

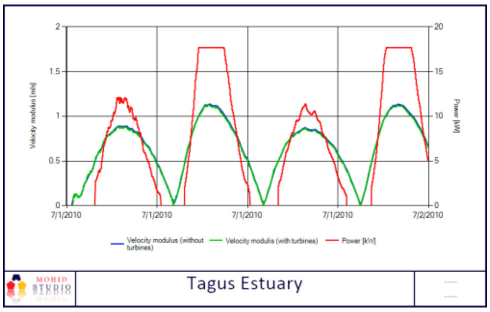

*Figure 11. Power and velocity output of one group of 10 turbines. Source: MOHID Studio*

## 6. Conclusions

A good compromise between the computational cost and the results have been achieved. The implementations capabilities are quite good and allow some interesting studies of energy extraction, array layout and flow modification. This implementation is a reliable and simple model of the effect of turbines inside a realistic and complex three dimensional hydrodynamic model.

Nevertheless it has some limitations, the most important of them is the non-discretization in the horizontal plane. The implementation is inappropriate to work with really large sizes of grid cells as the visual effects are not accurate. Also it cannot work with high resolutions where the grid cell side is smaller than the turbine diameter as the results will be incorrect, due to the non-horizontal discretization.

## 7. Future work

There are three main lines of action in order to continue with the project presented in this thesis. First of all test the implementation with more complex simulations. Test it in real environments or make studies of the impact of turbines in transport sedimentation are some ideas in order to squeeze the potential of the implementation and prove its reliability.

The second will be to improve the input data format in order to provide a more accessible tool if the user want to study huge arrays of turbines.

The third one will be the development of a horizontal discretization in order to surpass the limitation of the model so it can became more realistic. This last implementation can increase the computational cost of the simulations.

## References

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

[2] International Energy Agency. (2017). Electricity information: Overview. Retrieved from: http://www.iea.org/publications/freepublications/publication/ElectricityInformation2017Overview.pdf

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

[4] David R. Plew, Craig L. Stevens

Numerical modelling of the effect of turbines on currents in a tidal channel – Tory Channel, New Zeland (2013)

[5] 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.

[6] 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