

# 2D Fluid-Structure Interaction analysis of a vertical axis tidal turbine

Pol Gomà

November 18, 2016

The main purpose of this short presentation is to give a general overview of the project I worked on during the last semester of my degree, hence my final degree project. In said project I carried out a 2 dimensional Fluid-Structure Interaction (FSI) analysis of a vertical axis tidal turbine using both Ansys Fluent and Ansys Mechanical. Since the maximum time allowed in this presentation was of just 2 minutes, an utterly brief introduction to the project has to be done.

The first thing that is explained is the simplifications carried out on the geometry. Since a 3D FSI analysis would be of the utmost computational expensiveness, a cross section of the turbine was chosen as the domain of analysis. This way, the pressure and velocity fields were computed through a computational fluid dynamics simulation using Ansys Fluent. This simulation was actually more complicated than what one might think, and this is due to the fact that the ring containing the airfoils of the turbine has an angular velocity, hence it rotates. Consequently, a technique called *The Sliding Mesh technique* was used, which consists in a transient calculation in which some parts of the domain are actually moved each time step, leading to the mesh changing with time. Once a steady state solution was achieved, the fluid loadings were imported to Ansys Mechanical in order to compute the deformation of the blades, carrying out what is called a 1-way FSI analysis. To conclude, the deformation of the blades was computed for different materials with the purpose of identifying which one was more suitable for this kind of devices.