NAFEMS UK Regional Conference 2018 - Abstract Submission

Submission Date	2018-02-02 04:19:03
Name	Mr. Anthony Mosquera
Job Title	Design Services Manager
Company	Anthony Mosquera
Please identify the event for which your submitting?	NAFEMS UK Conference 2018
Will you be the presenting author?	No
Presenting Author	Mr. Daniel Hung
Presenting Author Company	Applied Computing & Engineering Ltd
Presenting Author Job Title	Senior Analyst
Presenting Author City, State/Province	Daresbury United Kingdom
Presentation Title	Fluid-Structure Interaction of a rigid wing for Minesto Deep Green, a tidal energy device
Relevant Themes / Keywords	Fluid Structure Interaction, FSI, CFD, modal analysis, renewable energy, tidal flow

Abstract (plain text)	The EU-project “Powerkite” is developing a next generation power take-off system (PTO) for a novel tidal energy collector concept, the Minesto Deep Green subsea tidal kite. The Minesto Deep Green concept is to extract tidal energy by ’flying’ a turbine through water. The primary forces for driving the turbine and associated kite are provided by a rigid wing and rudder system which is controlled so that the whole system flies in a figure-of-eight trajectory at many times the tidal flow velocity. Although only 12m in span, the lift forces on the main kite wing could be sufficient to lift a jumbo jet and distortion of the wing under such loads could adversely affect its hydrodynamic performance. Overall buoyancy considerations for the kite mean that the wing must be light and so the structure is assembled from composite elements that must provide the necessary stiffness to maintain the wing shape. The paper will present fluid-structure interaction analysis using two methods of different fidelity and methodology: firstly, an iterative method using relatively low fidelity tools which represent the wing structure as a series of beams which deflect and rotate under hydrodynamic loads, computed using potential flow (panel) methods and secondly, a sequential method that performs a 3D structural modal analysis, the results of which are used in a 3D CFD simulation that couples the modal equations with the Navier-Stokes equations to compute the deformed shape of the wing under hydrodynamic load. In the first method, the hydrodynamic loads are provided to the structural solver which computes the wing deflections and these are then passed back to the fluids solver. Revised hydrodynamic loads are computed and passed back to the structural solver. This iterative processes is repeated until there is no further change in the shape of the wing and the computed loads. The second method completely separates the initial structural analysis from the CFD analysis and no iteration between the solvers is required. The geometry updates and remeshing of the wing geometry are both handled within the CFD software. This latter method provides great flexibility in the choice of structural analysis solver as no direct coupling between the softwares is required. Although applied quite successfully in other applications, some assumptions made in the first low fidelity method proved to be unfounded giving quite different results to those obtained with the higher fidelity method. ACKNOWLEDGMENT This project has received funding from the European Union’s Horizon 2020 research and innovation programme under grant agreement No 654438.
abstract id	UK18-33