Programme Area: Marine

Project: PerAWAT

Title: Rationalised Flow Field Modelling Report

Abstract:

This deliverable outlines the method adopted to evaluate the spatial flow at site location, for use within the TidalFarmer suite of tools. This report describes the requirements of the flow field model for this application, and reviews existing packages based on these requirements. The reasons for the selection of the preferred solution are given. Following on from this deliverable, testing of a working method through tool beta testing and the development of a protocol for interfacing with an existing shallow water solver were developed.

Context:

The Performance Assessment of Wave and Tidal Array Systems (PerAWaT) project, launched in October 2009 with £8m of ETI investment. The project delivered validated, commercial software tools capable of significantly reducing the levels of uncertainty associated with predicting the energy yield of major wave and tidal stream energy arrays. It also produced information that will help reduce commercial risk of future large scale wave and tidal array developments.

ETI Marine Programme Project
PerAWaT MA1003
WG3WP4 D4 RATIONALISED FLOW FIELD MODELLING REPORT

Client	**Energy Technologies Institute**
Contact	Geraldine Newton-Cross
Document No	104329/BR/04
Issue	2.0
Classification	Not to be disclosed other than in line with the terms of the Technology Contract
Date	3rd March 2011

Author: L Gill / M Thomson

Checked by: D McCowen

Approved by: R Rawlinson-Smith

Garrad Hassan & Partners Ltd

St Vincent's Works, Silverthorne Lane, Bristol BS2 0QD, UK Tel: +44 (0)117 972 9900, Registered in England 1878456

IMPORTANT NOTICE AND DISCLAIMER

1. This report ("Report") is prepared and issued by Garrad Hassan & Partners Ltd ("GH" or "Garrad Hassan") for the sole use of the client named on its title page (the "Client") on whose instructions it has been prepared, and who has entered into a written agreement directly with Garrad Hassan. Garrad Hassan's liability to the Client is set out in that agreement. Garrad Hassan shall have no liability to third parties (being persons other than the Client) in connection with this Report or for any use whatsoever by third parties of this Report unless the subject of a written agreement between Garrad Hassan and such third party. The Report may only be reproduced and circulated in accordance with the Document Classification and associated conditions stipulated or referred to in this Report and/or in Garrad Hassan's written agreement with the Client. No part of this Report may be disclosed in any public offering memorandum, prospectus or stock exchange listing, circular or announcement without the express written consent of Garrad Hassan. A Document Classification permitting the Client to redistribute this Report shall not thereby imply that Garrad Hassan has any liability to any recipient other than the Client.
2. This report has been produced from information relating to dates and periods referred to in this report. The report does not imply that any information is not subject to change.

KEY TO DOCUMENT CLASSIFICATION

Strictly Confidential	:	For disclosure only to named individuals within the Client's organisation.
Private and Confidential	:	For disclosure only to individuals directly concerned with the subject matter of the Report within the Client's organisation.
Commercial in Confidence	:	Not to be disclosed outside the Client's organisation
GH only	:	Not to be disclosed to non GH staff
Client's Discretion	:	Distribution for information only at the discretion of the Client (subject to the above Important Notice and Disclaimer).
Published	:	Available for information only to the general public (subject to the above Important Notice and Disclaimer and Disclaimer).

REVISION HISTORY

Issue	Issue date	Summary
1.0	13/12/10	Original issue (electronic version only)
2.0	03/03/11	Second release, revised according to ETI feedback (electronic copy only)

CONTENTS

LIST OF TABLES

LIST OF FIGURES

EXECUTIVE SUMMARY

This document outlines the method adopted to evaluate the spatial flow field at a site location, for use within the TidalFarmer suite of tools. GH will not develop its own flow solver, but will instead develop links with existing codes. The model will provide the 2-d spatial variation of the flow speed and direction across the site, and this will be developed into 3-d in the vicinity of the array using standard boundary layer theory. Perturbations will be applied to the flow field through the wake and blockage models, to determine the effects of energy extraction.

This report describes the requirements of the flow field model for this application, and reviews existing packages based on these requirements. The reasons for the selection of the preferred solution are given. GH’s general modelling philosophy and the approach to this specific task is discussed, followed by the justification for this method and the theory and methodology behind it.

The method of integration within TidalFarmer is discussed briefly. However, the model will be run externally to the base module of the TidalFarmer tool, and hence the level of integration required is minimal.

SUMMARY OF NOTATION

Turbine characteristics & constants

b	Viscous drag coefficient
d_h	Hydraulic diameter (m)
f	Darcy friction factor
F	Coriolis coefficient associated with the Coriolis force
g	Acceleration due to gravity
h	Height above seabed (m)
H	Mean height of the horizontal pressure surface (e.g. free surface) (m)
k_s	Roughness height of the seabed form (m)
K	von Karman constant (K = 0.4)
N	Velocity distribution exponent
Q	Angular rotation rate of the Earth (π/12 radians/hour)
u	Velocity in the x direction (m/s)
u_h	Velocity in the x direction at height h (m/s)
U	Current speed at sea surface (m/s)
V	Velocity in the y direction (m/s)
η	Deviation of the horizontal pressure surface (e.g. free surface) from its mean
φ	Latitude
θ	Bed slope

Abbreviations

2-d	two dimensions
3-d	three dimensions
ADCIRC	ADvanced CIRCulation model
ADP	Acoustic Doppler Profiler
AMM	Atlantic Margin Model
CFD	Computational fluid dynamics
GPL	General Public Licence
HRCS	High Resolution Continental Shelf
IEC	International Electrotechnical Commission
LGPL	Lesser General Public Licence
MPI	Message Passing Interface
NIWA	National Institute of Water and Atmospheric Research
POL	Proudman Oceanographic Laboratory
ROMS	Regional Ocean Model System
SCRUM	S-coordinate Rutgers University Model
UKCS	UK Continental Shelf

A general glossary on tidal energy terms was provided as part of WG0 D2 – "Glossary of PerAWaT terms". This is a working document which will be revised as the project progresses.

1 INTRODUCTION

1.1 Scope of this document

This document constitutes the fourth deliverable (D4) of working group 3, work package 4 (WG3WP4) of the PerAWaT (Performance Assessment of Wave and Tidal Arrays) project funded by the Energy Technologies Institute (ETI). Garrad Hassan and Partners Ltd (GH) is the sole contributor to this work package. This document describes the theory behind and the method of implementation of the mathematical models used to evaluate the flow field in the proposed location of a tidal stream turbine or array of tidal stream turbines.

1.2 Purpose of this document

The purpose of WG3WP4 is to develop, validate and document an engineering tool that allows a rapid assessment of the energy yield potential of a tidal turbine array on non-specialist hardware. The specific objective of WG3WP4 D4 is to both document and provide a technical justification for the use of the flow field model within the suite of models that make up the engineering tool 'TidalFarmer'.

1.3 Specific tasks associated with WG3 WP4 D4

WG3WP4 D4 comprises the following aspects:

- A technical justification for the rationalised modelling approach and a description of the theory including fundamental assumptions
- A description of the modelling methodology
- A description of the method of integrating the flow field model into the TidalFarmer code

1.4 WG3 WP4 D4 acceptance criteria

The acceptance criteria as stated in Schedule 5 of the PerAWaT technology contract are as follows:

D4: Rationalised flow field modelling report describes:

- The theory and methodology (assumptions and algorithms) behind the rationalised flow field model, including spatial variation and depth profile.
- The method of integrating this model within the Beta code.

2 BACKGROUND AND RELEVANT THEORY

2.1 Introduction

This document is concerned with the modelling of the flow field in the proposed location of a tidal stream turbine / array (hereafter assumed to be an array), but excluding the effects of the array itself. It will feed into the TidalFarmer tool by providing the flow field to which the array will be added, and upon which the blockage and wake models will act. The flow solver will characterise the flow at basin scale, but will provide sufficient resolution in the array location for detailed array design.

The figure below illustrates how the outputs from the flow solver (site flow field) fit within the overall TidalFarmer tool.

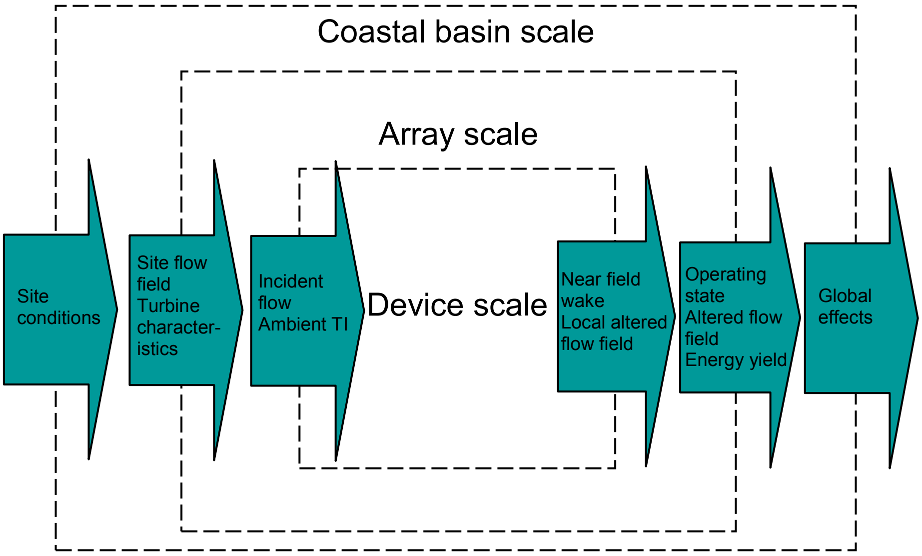

Figure 2-1: Hierarchy of modelling domains and scales

2.2 Characterisation of tidal models

Models of tidal flows can generally be categorised by how they deal with temporal and spatial variations in the flow field. The time domain can be dealt with either as a time-stepping model or a frequency (harmonic) model, and models will either represent the space domain using a structured or unstructured grid.

Whilst harmonic tidal models are computationally-efficient, they are limited in their representation of the flow structure (i.e. tidal eddies) in the strongly advective local flows usually present at sites with strong tides. They are therefore only recommended by the draft 'Tidal energy resource characterisation and assessment' IEC technical specification[1] for use during site scoping exercises or to provide model boundary conditions for time-stepping models, and are not recommended for the analysis of detailed tidal flows within an array.

To evaluate the variation in flow across a space, either time varying or steady models which calculate the impact of the local geography are required. Both the seabed form (bathymetry) and roughness influence a tidal flow, causing it to vary across a site. The governing equations which can be used to describe fluid flows are the Navier-Stokes equations. These equations are routinely rationalised to allow understanding and a much reduced effort in deriving a solution. For the analysis of coastal basin flows, the Navier-Stokes equations are rationalised to the shallow water equations which solve for a depth-averaged 2-d flow field.

2.3 The shallow water equations

The shallow water equations, also called the Saint-Venant equations in their unidirectional form, are a set of hyperbolic, partial differential equations which describe the flow below a pressure surface (for example a free surface) in a fluid. They are widely applicable, and are used in areas as varied as the calculation of tides and storm waves, the impact and stability of structures, sedimentology, the simulation of dam-break flood waves, the study of floods, etc.

The shallow water equations are derived by depth-integrating the Navier-Stokes equations in cases where the vertical velocity of the fluid can be assumed to be small. This can be assumed in situations where the horizontal length scale is much greater than the vertical length scale; under this condition the conservation of mass implies that the vertical velocity of the fluid is small. The momentum equation shows that the vertical pressure gradients are nearly hydrostatic, and that horizontal pressure gradients are due to the displacement of the pressure surface. This implies that the horizontal velocity field is constant throughout the depth of the fluid.

The shallow water equations are based on a number of key fundamental assumptions, namely:

- The streamline curvature is very small and the vertical fluid accelerations are negligible, leading to a hydrostatic pressure distributions
- The flow resistance and turbulent losses are the same as for a steady uniform flow for the same depth and velocity, regardless of trends of the depth
- The bed slope, θ, is sufficiently small to be approximated by $\cos\theta \approx 1$ and $\sin\theta \approx \tan\theta$
- The water density is constant

With these basic hypotheses, it is possible to describe the flow at any point in time by two variables; flow velocity and water depth.

The above assumptions defining the validity of the shallow water equations are unlikely to be true for real sites; however as stated above, the equations are widely used for the modelling of a variety of sites and applications. The shallow water models which use these equations are typically subject to high degrees of spatial uncertainty due to the non-ideal nature of the sites. This does not restrict their use, but the uncertainty associated with the results must be taken into account. This can be seen by the types of applications commonly modelled with the shallow water equations, including sediment transport (where the focus is on overall transport rather than the exact flow at specific locations) and the modelling of extreme flows (where large safety factor are typically added). GH's method of reducing uncertainty, by relating the modelled outputs to actual site measurements, is described in detail in Section 4.2.

The shallow water equations can be expressed in their conservative (in terms of momentum) or non-conservative (in terms of velocities) forms.

Conservative form:

(in the case of no Coriolis, frictional or viscous forces):

$$\frac{\partial \eta}{\partial t} + \frac{\partial(\eta u)}{\partial x} + \frac{\partial(\eta v)}{\partial y} = 0$$

$$\frac{\partial(\eta u)}{\partial t} + \frac{\partial}{\partial x}\left(\eta u^2 + \frac{1}{2} g \eta^2\right) + \frac{\partial(\eta u v)}{\partial y} = 0$$

$$\frac{\partial(\eta v)}{\partial t} + \frac{\partial(\eta u v)}{\partial x} + \frac{\partial}{\partial y}\left(\eta v^2 + \frac{1}{2} g \eta^2\right) = 0$$

Non-conservative form:

$$\frac{Du}{Dt} - F\, v = -g \frac{\partial \eta}{\partial x} - bu$$

$$\frac{Dv}{Dt} - F\, u = -g \frac{\partial \eta}{\partial y} - bv$$

$$\frac{\partial \eta}{\partial t} = -\frac{\partial}{\partial x}\left(u\,(H + \eta)\right) - \frac{\partial}{\partial y}\left(v\,(H + \eta)\right)$$

where $(F = 2Q \sin \varphi)$ on Earth

A derivation of the shallow water equations is given in Appendix 1.

The vertical velocity term is removed from the Navier-Stokes equations to form the shallow water equations, but it can be recovered via the continuity equation once a solution has been found, namely that the horizontal velocities and free surface displacement have been determined. This is important because although the vertical velocities have been removed from the equations, they are not necessarily zero (e.g. the vertical velocity will not be zero when changes of depth occur).

2.4 Solving the shallow water equations

As a set of hyperbolic, partial differential equations, the shallow water equations require numerical methods to derive a solution to the bounded problem. The shallow water equations are typically solved using finite difference, finite element, or finite volume numerical methods.

Structured grid models usually use a finite difference formulation, and normally have a fixed spatial discretisation. For models covering a large area this can lead to either very long run times, or a lack of detail in the area of interest. Finer scale sub-models are therefore typically nested within the larger model to increase resolution where required. Unstructured grid models present a solution to this issue by allowing variable grid sizes, allowing large grids at the model boundaries while achieving high resolution in the area of interest. Unstructured grid models typically use finite element and/or finite volume formulation methods. Both structured and unstructured grid models can be used to model the detailed flow conditions at tidal sites, provided they can accurately represent the flow structure in the strongly advective flows typical at these sites[1].

The shallow water equations represent the 2-d flow field only, and hence models based on these equations will not directly represent factors which vary with depth. However, multiple sets of shallow water equations can be used to model a number of ‘layers’ of the flow in cases where the mean state is sufficiently simple. This allows the vertical variations to be dealt with separately to the horizontal variations. This is a method employed by many of the apparently 3-d modelling packages, for example TELEMAC-3D. These models do not solve the 3-d system of Reynolds Averaged Navier-Stokes equations (RANSE), but instead they solve for multiple layers of 2-d flows coupled by the 3-d continuity equation. As a result these models are more commonly referred to as 2.5-d models.

The computational effort required to solve the 2.5-d set increases directly with the number of layers. The UK continental shelf model (used for the Atlas of UK Marine Renewable Energy Resources[2]) uses 34 layers, and typically a minimum of a layer per meter is required to pick up variations in the depth profile due to changes in bathymetry and seabed roughness. The common view of these models is that they represent changes to the boundary conditions due to large bathymetry changes.

RANSE solvers provide a solution to the 3-d flow field by solving the 3-d system of equations directly. However, the size of the computational domain required to capture the tidal dynamics requires dedicated computational resources. [3, 4, 5]

3 REVIEW OF EXISTING APPROACHES TO FLOW FIELD MODELLING

Numerous flow field modelling packages are available, from open-source code to commercial packages. With a wide variety of modelling options available, GH will not be developing its own in-house spatial flow field model, but will instead interface with existing models.

This section outlines the basic requirements of the flow field model and then describes a number of modelling packages. The various modelling packages are then compared, reviewing the relevant advantages and disadvantages within the context of their applicability to the stated requirements.

3.1 Flow field model requirements

The purpose of the flow field model is to predict the spatial flow field variation within a tidal array site due to the seabed form.

The model must be able to represent the flow field to a reasonable resolution. The draft ‘Tidal energy resource characterisation and assessment’ IEC technical specification[1] specifies a minimum requirement for grid resolution of 50m within the array site, but a relative value dependent on the size of the device is of greater relevance than an absolute value. A resolution of 0.5 – 2 turbine diameters is considered by GH to be the minimum required to sufficiently represent the flow conditions through the array. A resolution of 2 turbine diameters will provide a reasonable indication of the flow conditions across the array, however this assumes a linear gradient across the rotor swept area and thus may not be an accurate representation the flow conditions. 2 turbine diameter is therefore considered by GH to be the minimum acceptable resolution. A resolution of 0.5 turbine diameters is considered to be the ideal situation, providing greater resolution across the rotor to enable a more accurate calculation to be made of the current speeds, and hence the energy yield, across the rotor swept area.

Although this requirement for high resolution does not in itself exclude the use of a structured grid model (e.g. ROMS), the use of an unstructured grid model will allow for a more computationally-efficient solution. The ability to vary the grid size allows the user to use a large grid at the model boundaries, along with a finer discretisation of the model within the tidal array site.

The model must evaluate the variation in flow across the site in order to access the impact of device location on the energy yield of a tidal array. The site and surrounding area are typically analysed using 2-d models rather than the much more computationally intensive 3-d flow field models. However, to accurately evaluate the impact of depth varying flow incident on a rotor the 2-d depth-averaged flows will need to be developed into a 3-d flow field. Further discussion of 2-d versus 2.5-d models is given in the following section, and the GH approach to developing a 3-d flow field is given in Section 4.3. It is also noted that a comparison of 2-d and 3-d coastal basin models will be made in WG3 WP2, WP3 & WP6.

A number of the models have open-source code, and although this is not a necessary requirement, it would be a beneficial addition. The ability to work with the code of the modelling software will allow GH to make amendments as necessary as well as to better integrate it into the TidalFarmer programme.

A further consideration is the ability of the TidalFarmer tool to interface with the flow field modelling package from a conventional user platform such as a desktop computer / laptop.

3.2 Modelling methodologies

As noted above, this section will discuss and compare a number of shallow water codes. This list is not intended to be exhaustive, but instead represents a list of models which could be suitable for the above-mentioned purpose.

Table 3-1: Comparison of flow field modelling software

Model	Detail	
TELEMAC-2D TELEMAC-3D	Description	2-d and 2.5-d open channel flow models. TELEMAC solves the shallow water equations, either vertically-averaged in two dimensions or layered in three dimensions. TELEMAC-2D solves the Saint-Venant equations using the finite-element or finite-volume method and a computation mesh of triangular elements, and TELEMAC-3D solves the Navier-Stokes equations to simulate tri-dimensional flow. Space is discretised in the form of an unstructured grid of triangular elements, which means that it can be refined particularly in areas of special interest as well as allowing realistic representations of complicated coastlines and bathymetries. TELEMAC includes horizontal turbulence options for the simulation of very detailed flow patterns, spherical co-ordinates for very large area models, simulation of wetting and drying within the model domain and solution for transcritical flow. TELEMAC-3D can also simulate three dimensional flow affected by stratification (thermal or saline), wind or wave breaking. Turbulence models available include k-epsilon and mixing length. TELEMAC-2D can perform simulations in transient and permanent conditions. It can take into account: – Bed friction – Influence of Coriolis force – Influence of meteorological factors: atmospheric pressure and wind – Turbulence – Torrent and river flows – Influence of horizontal temperature or salinity gradients on density – Cartesian or spherical coordinates for large domains – Dry areas in the computational domain: intertidal flats and flood plains – Monitoring of floats and Lagrangian drifts – Treatment of singular points: sills, dikes, pipes

		TELEMAC-2D offers the user a set of FORTRAN sub-routines that can be modified to meet the specific requirements of each model: specification of initial conditions or complex boundary conditions, link-ups with other modelling systems, introduction of new functions.[6,7,8,9,10]
	Dimensions	2-d / 2.5-d
	Grid discretisation	Unstructured grid
	Code availability and support availability	TELEMAC-2D: Free-ware and open-source code – The TELEMAC-2D code can be used as an external add-on to a commercial product, but it is not permissible to use the code directly within a commercial product ("All freeware TELEMAC is distributed under GNU General Public Licence (GPL), except the library BIEF which is given under Lesser GNU General Public Licence (LGPL)"[11]) TELEMAC-3D: Share-ware (free licence executables, but maintenance licence required) User support provided.
	Equipment required	Standard computer with required software
MIKE 21 (HD) MIKE 3 (HD)	Description	2-d and 3-d open channel flow. MIKE 21 and MIKE3 are 2-d and 3-d (respectively) modelling tools from DHI to simulate flow phenomena and related processes in coastal areas and seas. Various modules are available, including MIKE21 HD and MIKE3 HD (Hydrodynamics). MIKE 21 HD and MIKE3 HD simulate the water level variation and flow in response to a variety of forcing functions in lakes, estuaries, bays and coastal areas. The water levels and flows are resolved on either a rectilinear grid, a curvilinear grid, a triangular element mesh or any combination hereof covering the area of interest. In addition, the MIKE 21 & MIKE 3 Flow Model FM is a new general hydrodynamic flow modelling system based on a finite volume method on an unstructured mesh. It is used to examine complex free surface flow regimes where the limitations of rectilinear or curvilinear grids require an unstructured grid approach. The spatial discretisation of the governing equations is performed using a cell-centred finite difference method. In the horizontal plane an unstructured grid is used while a structured mesh is used in the vertical domain (3-d).

		MIKE 21 / 3 HD include formulations for the effects of: – convective and cross momentum – bottom shear stress – wind shear stress at the surface – barometric pressure gradients – Coriolis forces – momentum dispersion (through e.g. the Smagorinsky formulation (MIKE 21 / 3 HD) or the k-e model (MIKE 3 HD)) – wave-induced currents (MIKE 21 HD) – density effects (MIKE 3 HD) – sources and sinks (mass and momentum) – evaporation and precipitation – flooding and drying – hydraulic structures Hydrographic boundary conditions can be specified as a constant or variable (in time and space) level or flux (MIKE 21 HD) / velocity (MIKE 3 HD) at each open model boundary, as a constant or variable source or sink anywhere within the model, and as an initial free surface level map applied over the entire model. (3-d).[12,13,10]
	Dimensions	2-d, 3-d
	Grid discretisation	Unstructured (FM model) / structured (non-FM model)
	Code availability and support availability	Commercial product (not open-source). User support provided.
	Equipment required	Standard computer with required software
ANSYS Fluent ANSYS CFX	Description	Computational fluid dynamics (CFD) packages. ANSYS Fluent and ANSYS CFX are the main general-purpose fluid simulation products offered by ANSYS. These two solvers were developed independently over decades and have a number of things in common, but they also have some significant differences. Both are control volume-based for high

		accuracy and rely heavily on a pressure-based solution technique for broad applicability. The products differ mainly in the way they integrate fluid flow equations and in their equation solution strategies. The ANSYS CFX solver uses finite elements (cell vertex numerics), similar to those used in structural analysis, to discretise the domain. In contrast, the ANSYS FLUENT solver uses finite volumes (cell-centred numerics). Ultimately, though, both approaches form control volume equations that ensure exact conservation of flow quantities, a vital property for accurate CFD simulations. ANSYS CFX focuses on one approach to solve the governing equations of motion (coupled algebraic multigrid), while ANSYS FLUENT offers several solution approaches (density-based as well as segregated and coupled pressure-based methods). Combinations of elements in a variety of shapes are permitted, such as quadrilaterals and triangles for 2-D simulations and hexahedra, tetrahedra, polyhedra, prisms and pyramids for 3-D simulations.[14,15]
	Dimensions	2-d / 3-d
	Grid discretisation	Unstructured
	Code availability and support availability	Commercial product. User support provided.
	Equipment required	High specification computer with required software.
STAR-CD	Description	Computational fluid dynamics (CFD) packages. STAR-CD V4 is a general-purpose commercial finite-volume code that performs Computational Continuum Mechanics simulations. Both fluid and solid calculations are performed simultaneously on a single computational mesh, created automatically with CD-adapco's automatic meshing technology. The mesh (which can be constructed from hexahedra, tetrahedra or arbitrary polyhedra) automatically represents the interface between different material regions (whether fluid-solid or solid-solid) using a conformal interface, which means that solution domains are coupled implicitly without mapping or interpolation. STAR-CD provides a range of physics modelling, capable of simulating engineering problems that involve turbulence, combustion, heat transfer (including radiation heat transfer), reacting flows and

		multiphase physics. [16]
	Dimensions	2-d / 3-d
	Grid discretisation	Unstructured
	Code availability and support availability	Commercial product. User support provided.
	Equipment required	High specification computer with required software.
Gerris	Description	Gerris is a free software program for the solution of the partial differential equations describing fluid flow. Gerris uses a quadtree (octree in 3-d) finite volume discretisation. A brief summary of its main features: – Solves the time-dependent incompressible variable-density Euler, Stokes or Navier-Stokes equations – Solves the linear and non-linear shallow-water equations – Adaptive mesh refinement: the resolution is adapted dynamically to the features of the flow – Entirely automatic mesh generation in complex geometries – Second-order in space and time – Unlimited number of advected/diffused passive tracers – Flexible specification of additional source terms – Portable parallel support using the MPI library, dynamic load-balancing, parallel offline visualisation

		– Volume of fluid advection scheme for interfacial flows – Accurate surface tension model. [17]
	Dimensions	2-d / 3-d
	Grid discretisation	Unstructured
	Code availability and support availability	Open-source code: the source code is available free of charge under the Free Software General Public Licence (GPL). The Gerris software is supported by NIWA (National Institute of Water and Atmospheric Research) and Institut Jean le Rond d'Alembert, but limited user support is available
	Equipment required	Standard computer with required software
ROMS	Description	The Regional Ocean Model System (ROMS) is a free-surface, hydrostatic, primitive equation ocean model that uses stretched, terrain-following coordinates in the vertical and orthogonal curvilinear coordinates in the horizontal. Initially, it was based on the S-coordinate Rutgers University Model (SCRUM) described by Song and Haidvogel (1994). ROMS was completely rewritten to improve both its numerics and efficiency in single and multi-threaded computer architectures. It was also expanded to include a variety of new features including high-order advection schemes; accurate pressure gradient algorithms; several subgrid-scale parameterizations; atmospheric, oceanic, and benthic boundary layers; biological modules; radiation boundary conditions; and data assimilation.[18,19,10]
	Dimensions	2-d / 2.5-d
	Grid discretisation	Structured grid: orthogonal curvilinear grids
	Code availability and support availability	Open-source code. Limited user support available
	Equipment required	Standard computer with required software
Delft3D	Description	Delft3D is a 2-d/2.5-d modelling suite to investigate hydrodynamics, sediment transport and morphology and water quality for fluvial, estuarine and coastal environments. The FLOW module is a multi-dimensional (2-d or 2.5-d) hydrodynamic (and transport) simulation program which calculates non-steady flow and transport phenomena resulting from tidal and

		meteorological forcing on a curvilinear, boundary-fitted grid. In 3-d simulations, the vertical grid is defined following the so-called sigma coordinate approach. The constant number of vertical layers over the whole of the computational field used in this approach results in a high computing efficiency. The RGFGRID module allows the generation of orthogonal, curvilinear grids of variable grid size for the hydrodynamic module Delft3D FLOW. Standard features include: – Coriolis force – Advection-diffusion solver included to compute e.g. density gradients (due to non-uniform temperature and salinity concentration distributions) – Inclusion of pressure gradients terms in the momentum equation (density driven flows) – Turbulence model to account for the vertical turbulent viscosity and diffusivity based on the eddy viscosity concept. Four options: k-epsilon, k-L, algebraic and constant model are provided – Shear stresses exerted by the turbulent flow on the bottom based on a quadratic Chézy or Manning's formula – Simulation of drying and flooding of inter-tidal flats (moving boundaries) for both 2-d and 2.5-d cases Special features include: – Various options for the co-ordinate system (rectilinear, curvilinear and spherical) – Built in automatic switch converting 2-d bottom-stress coefficient to 2.5-d coefficient – Horizontal turbulent exchange coefficients as the sum of a 2-d turbulence model and a 2-d sub-grid turbulence model – Optional facility for tidal analysis: output parameters (water elevation and/or velocities) can be analysed in terms of tidal components using the program TRIANA. The results from these analyses can be subsequently compared with observation data supplied by the user.[20, 10]
	Dimensions	2-d / 2.5-d
	Grid discretisation	Unstructured (with RGFGRID module) / structured (without RGFGRID module)
	Code availability and support availability	Commercial product (not open-source). User support provided.

	Equipment required	Standard computer with required software

ADCIRC	Description	ADvanced CIRCulation model (ADCIRC).
		
		ADCIRC is a system of computer programs for solving time dependent, free surface circulation and transport problems in two and three dimensions. These programs utilize the finite element method in space allowing the use of highly flexible, unstructured grids.[21]
		
		Features and applications include:
		– A general purpose, explicit, time dependant coastal circulation model
		– Highly scalable model - from small scale coastal and estuarine environments to large scale open ocean models
		– Multiple boundary condition and forcing options including:
		    – Specified elevation
		    – Specified normal flow
		    – Tidal potential & forcing frequencies
		    – Wind/wave surface stress
		    – Atmospheric pressure
		– Extract tidal data from the global LeProvost or regional ADCIRC databases
		– Coupling with near shore wave models such as CMSWAVE and STWAVE [22,23]

	Dimensions	2-d / 3-d
	Grid discretisation	Unstructured
	Code availability and support availability	Commercial product. Limited user support available.
	Equipment required	Standard computer with required software
RMA10	Description	RMA10 is a dynamic three-dimensional finite element hydrodynamic model for computing water surface elevations and horizontal velocity components for stratified, free-surface flow. It is designed for water bodies in which vertical accelerations can be considered negligible (hydrostatic). The model solves the

		transport equation for salinity, temperature, and suspended sediment and incorporates the effect of these quantities on density. The model can incorporate 1-dimensional, 2-dimensional (laterally- or depth-averaged) and 3-dimensional elements into a single mesh as required by the application. RMA10 uses the Reynolds form of the Navier-Stokes equations; vertical turbulence is estimated with either a quadratic parameterization of turbulent exchange or a Mellor-Yamade Level 2 turbulence sub-model. RMA10 can simulate wetting and drying and marsh porosity and is therefore suited to computing hydrodynamics in tidal flats and wetlands. One version of RMA10 is a component of the US Army Corps of Engineers TABS-MD System.[24]
	Dimensions	3-d
	Grid discretisation	Unstructured
	Code availability and support availability	Proprietary code. Limited technical support: The USACE Waterways Experiment Station distributes the model and provides technical support only to USACE and Research Lab users.
	Equipment required	Standard computer with required software

3.3 Comparison of modelling software

The table below summarises the capabilities of the modelling packages considered above.

Table 3-2: Comparison of modelling software

Model	2-d	Unstructured grid	Open-source code	Sufficient user support	Standard equipment sufficient
TELEMAC-2D / 3D	*	*	*	*	*
MIKE 21 / 3 (HD)	*	*		*	*
ANSYS Fluent	*	*		*	
STAR-CD	*	*		*	
Gerris	*	*	*		*
ROMS	*		*		*
Delft3D	*	*		*	*
ADCIRC	*	*			*
RMA10		*			*

TELEMAC is the only code which fulfils all of the stated requirements. As a widely accepted tool for the modelling of tidal flows, TELEMAC provides a thoroughly developed and validated model to meet the requirements of a flow field solver.

The level of detail which can be modelled with RANSE solvers is greater than that possible with shallow water models. However, the computational requirements and model run times are also much greater, and this is inconsistent with the aim of developing a practical solution for developers in the iterative design process (see Section 4.1).

All but ROMS provide unstructured gridding capabilities, allowing for variable mesh sizes across the model domain.

It is the intention of the TidalFarmer software to interface with the results multiple flow solvers. This will allow the users of the TidalFarmer software to input their own choice of flow field, calculated via their preferred method. The GH software will pre-process the model results into the required form for inter-array modelling.

4 GH APPROACH TO FLOW FIELD MODELLING

4.1 GH modelling philosophy

GH's modelling philosophy is to provide engineering solutions to meet a commercial need, and in the case of tidal array design this means providing a design tool that can offer practical solutions to aid the iterative design process. To develop an appropriate design tool, rationalised modelling methods based on a physical understanding of the Navier-Stokes equations that provide robust estimates with known uncertainties are preferred to more complex numerical methods.

The development of a rationalised flow field modelling approach is based on the necessity to produce a detailed (fine numerical grid of the order of 0.5 rotor diameters) three-dimensional array-influenced velocity and turbulence field across the array site without overly burdensome computational effort. RANSE CFD solvers might be tuned for the analysis of an array once the layout is defined, but the effort required to iterate on the location of each device in order to optimise energy yield would be very large. Pseudo three-dimensional shallow water solvers are more computationally efficient than RANSE solvers, but are still considered too computationally expensive. On the other hand, depth-averaged models will not yield the required accuracy. These issues might be resolved with post-processing, however, the effort required to both pre- and post-process simulations for numerous array layouts would be considerable. The focus of the GH approach is to therefore to amalgamate the best of both methods.

4.2 Description of the GH approach to flow field modelling

The GH approach to flow field modelling is to interface with existing flow modelling package results to provide a relative spatial variation of flow speed and direction across the site, to which the effects of energy extraction (via the blockage and wake models) will be applied. A static link between the flow field and the energy extraction will be used, applying perturbations to the flow field created by the external modelling package and then verifying the results. Pre-processing within the TidalFarmer software will use both the provided flow modelling results and the available site data to yield a 3-d flow and turbulence intensity field.

To avoid excessive computational effort and provide a solution correlated to measured site data, the GH approach minimises shallow water model run time to a single representative tidal cycle. The calculated tidal cycle provides a spatial flow field which can then be correlated to long term site measurement data using the following approach. To allow a flexible and computationally-efficient inter-array analysis, the time stepping results from the flow solver model are converted to the frequency domain, i.e. the site reference data time series speeds are binned according to speed and direction. The spatial variations from the reference point are compiled to yield a grid of speed ups/downs for each speed bin and direction sector. These maps of flow variation will then be used together with the site data (bottom-mounted ADP data and/or vessel-mounted transect data) to create maps of speed-ups/slow-downs for each flow speed bin and direction sector relative to the reference points. In the vicinity of the array, a 3-d flow field will be generated by extrapolating the depth-averaged 2-d flow maps through the water column using shear profiles determined from the site data. The method for development of the 2-d flow field into 3-d is described in further detail in Section 4.4.

Figure 4-1 below illustrates the TidalFarmer method.

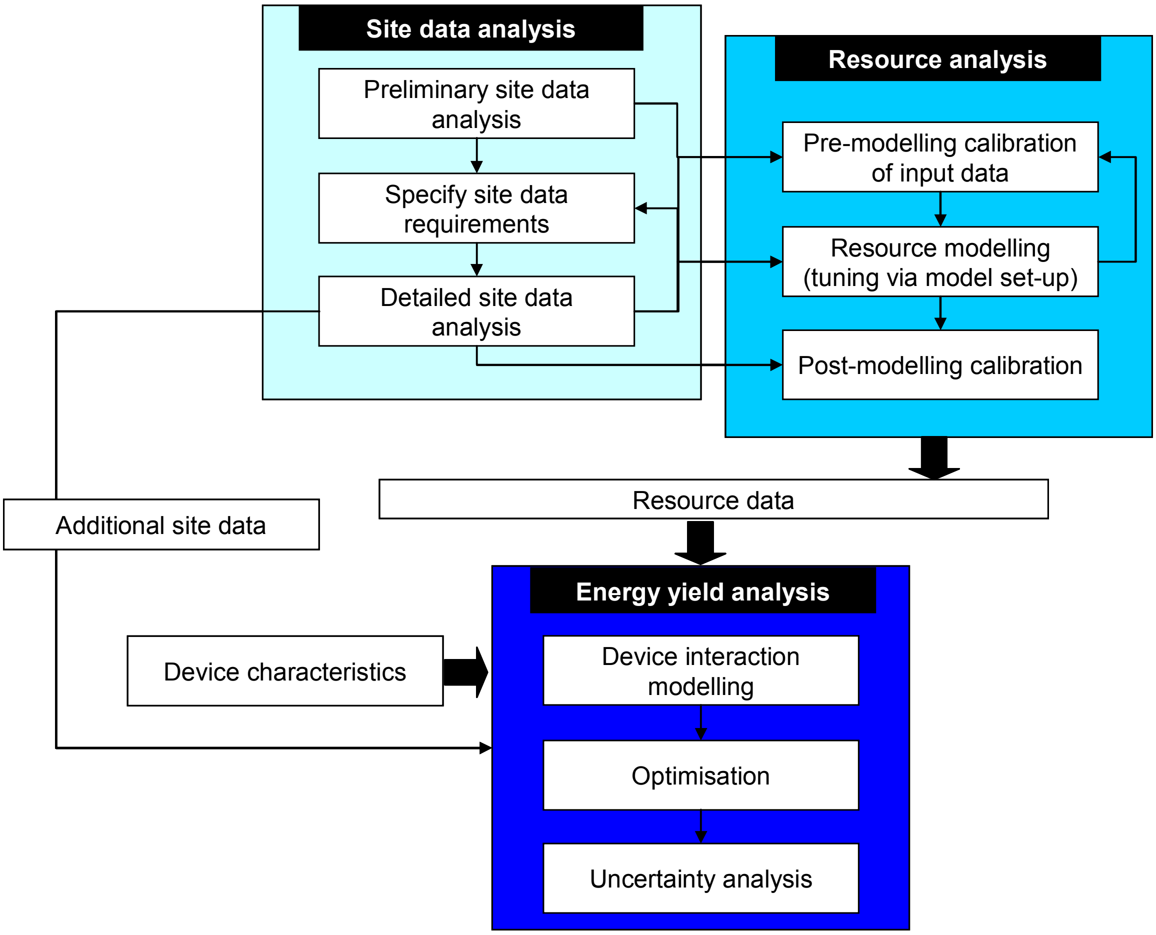

Figure 4-1: TidalFarmer method

The critical assumption in the overall GH approach is that the effect of energy extraction on the global flow is small and does not significantly alter the 2-d bathymetry-driven flow field (an example of a typical scenario is provided in Section 6.2 of WG3WP4 D3). Effectively this assumes that the momentum extracted by each turbine is small in comparison to the energy in the surrounding flow. There will be a reduction in the momentum due to the extraction of energy and thus a resulting drop in free-surface elevation downstream to accommodate this extraction. However, on a micro-scale the energy extracted by the turbine will be replenished from the across-stream flow and the effect on the downstream flow will be small.

This assumption does not mean that the inter-array effects are to be ignored but it does treat the array-influence on the flow field as a pertubration. Array-influenced modelling incorporates both hydrodynamic interaction between adjacent devices and the bounding surfaces of the channel (blockage modelling) and the wake effect upon the downstream flow (wake modelling). The development and recovery of the wake will be influenced by a number of variables including (a) the constraining boundaries of the sea bed and water surface, (b) features of the machine operation such as misalignment between the current direction and the machine axis, and (c) ambient and wave-generated turbulence in the current stream.

The driving mechanism behind the tidal stream resource is a gravitational head, as compared to a pressure (metrological) head in wind. Provided that the energy extracted by the array does not impinge on the driving head then the impact on the flow field is local. This assumption (i.e. that the array will remove a small amount of energy in comparison with the global flow) needs to be tested, either by conservative basic calculations (if the domain geometry is relatively simple e.g. channel connecting to seas) or with 2-d shallow water modelling for more complex geometry. In order to evaluate the impact of the array on the global flow field, a dynamic link is planned to connect the GH software with an existing flow solver. This will be further explained in the PerAWaT deliverable WG3WP4 D10 (Regional scale plug-in protocol report).

4.3 Justification for the GH approach to flow field modelling

For an energy yield analysis a 3-d flow field with a grid resolution of 0.5 – 2 turbine diameters will be required. Even with the use of unstructured grids, the model run times required to achieve such resolution is high. The number of design permutations is of the order of 1000s, and the need to iterate the array layout renders the direct solution approach (in which a representation of each turbine rotor in embedded within the flow solver) very computationally-expensive. A perturbation approach de-couples the flow solver from the inter-array flow modelling, and hence offers the potential for a much more computationally-efficient solution.

The fundamental assumption that the energy extracted by the array is small in comparison with the total energy in the system is based on both an understanding of the total energy in the tidal system, and from evidence where heavily-blocked channels (high ratios of swept area to channel cross-section) yield immeasurable changes to the downstream surface elevation. However, to check this assumption array representation within a basin scale model is required. To address this issue a dynamic link with a flow solver is required. However, re-running the flow solver for each array layout is not proposed, instead the approach uses an initial check and a post array-optimisation verification to check the global impact of an array on the flow field.

The development of a GH flow solver is not justified due to the extensive expertise and effort required to match the functionality of numerous existing shallow water modelling packages. A dynamic link with the multitude of different modelling packages is not feasible, however, the ability to use the spatial flow field model results is essential. The need to interface with existing flow solvers is on the basis that potential users will be using a variety of different model packages during the project development stage, and will therefore want to utilise existing models rather than developing new models in alterative software. This method allows for an efficient inter-array flow modelling approach but does not attempt to model the effect that the array has on the global flow field.

4.4 Development of a 3-d flow and turbulence field

The process of generating a 3-d flow field is dependant on the available data. Ideally, measured site data will be used directly or will validate a 2-d or 3-d shallow water model. Assuming there is some degree of confidence in a model's results (i.e. it correlates well with measured transect data) the GH approach is to first refine the 2-d or 3-d model's grid resolution by either re-running the existing model or by setting up a number of grid refinement steady flow simulations using the original model as the nested model's boundary conditions.

3-d flow field models can have excessive computing requirements (see Section 2), and hence typically 2-d depth averaged models are run. In order to generated the required 3-d flow field in the vicinity of the array the depth-averaged 2-d flow field will be extrapolated through the water column. A split power law model will be used to do this within the TidalFarmer software. A mixture of characterised site data (i.e. fitted power laws to the measured profile data) and open channel friction drag models will be used to evaluate the power law coefficients[3]. The power law function is:

$$\frac{u_h}{U} = \left(\frac{h}{H}\right)^N$$

For a uniform equilibrium flow in open channels

$$N = K\left(\frac{8}{f}\right)^{0.5}$$

where f is the Darcy friction function, and using Colebrook's formula for fully rough turbulent flow:

$$\left(\frac{1}{f}\right)^{0.5} = 2 \cdot \log_{10}\left(\frac{D_h}{k_s}\right) + 1.14$$

where D_h is the hydraulic diameter, and k_s is the roughness height of the seabed form.

Because there is little evidence validating these models in high-flowing tidal streams, the GH approach uses site-observed data to inform central power law coefficients for each flow speed and direction bin, and uses the relative results of the friction model to perturb the central value.

5 FLOW FIELD MODEL IMPLEMENTATION

Unlike the blockage and wake models (WG3WP4 D1 & D2), the flow field model will not be incorporated directly within the TidalFarmer software, and will operate externally to the base module. This is illustrated by Figure 5-1 below, which shows the envisaged structure of the TidalFarmer software.

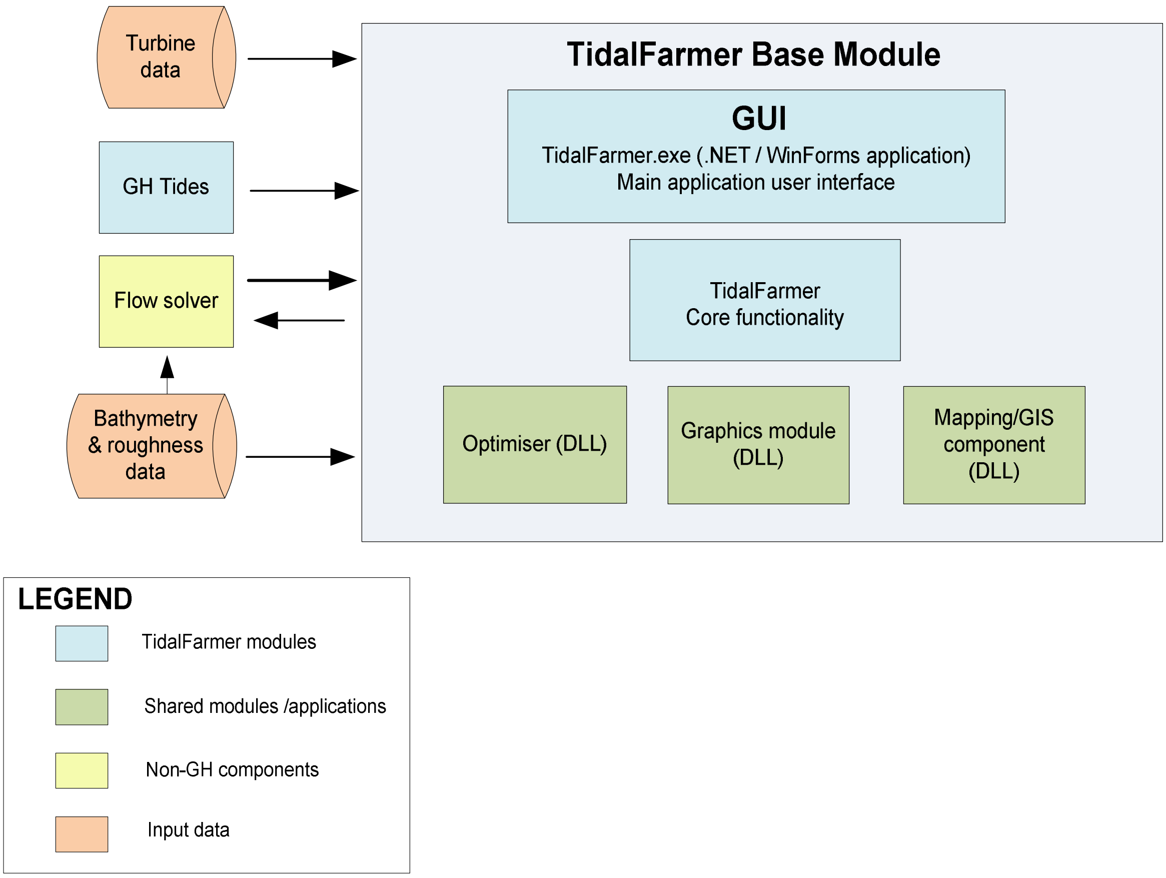

Figure 5-1: Summary of the TidalFarmer envisaged structure

The TidalFarmer software tool will consist of a single executable file (including a user interface) with which the user will interact, as well as a number of calculation modules. The tidal site data analysis tool (currently called 'GH Tides') will be implemented as a dynamic-link library (DLL), however the flow solver will be run as a separate executable file which will communicate by means of file exchanges.

As shown in the Figure 5-1, the link with the flow solver can be interactive with feedback from the GH software to the flow solver. The arrangement is referred to as a dynamic link. The protocol for a dynamic link and the interfacing options will be investigated in the PerAWaT deliverable WG3WP4 D10. As demonstrated in Table 3-2, TELEMAC satisfies all of the stated requirements for a flow field solver and is therefore the preferred software package to interface with the TidalFarmer software.

A dynamic link between the flow solver and TidalFarmer will be used pre and post array-optimisation, as mentioned in Section 4. However, the approach during optimisation will apply perturbations to the flow field and then assessing their impact on the energy extraction. The TidalFarmer software pre-processes the results of the 2-d or 3-d flow solver to produce the 3-d flow field for use in the inter-array modelling.

6 SUMMARY

This report describes the selection of the flow field model which will be used as part of the TidalFarmer software. The report discusses the existing approaches to shallow water modelling, and then identifies the preferred solution. The GH approach to flow field modelling is then discussed, and justification for this method is given along with the theory and methodology behind it. The method of integration within TidalFarmer is also discussed.

GH will not develop its own flow solver, but will instead use existing 2-d models (or 3-d models if available) and developing them into 3-d in the vicinity of the array. This will provide the spatial variation of the flow speed and direction across the site, upon which perturbations will be applied through the blockage and wake models to determine the effects of energy extraction. The model will be run externally to the base module of the TidalFarmer tool, and hence the level of integration required is minimal. A dynamic link between the flow solver and TidalFarmer will be used pre and post array-optimisation to assess the impact of global energy extraction.

To allow a flexible and computationally-efficient inter-array analysis, the temporal variations in flow is capture in the frequency domain via discrete speed and direction bins.

The next steps for this work package in relation to rationalised flow field modelling are:

- Testing of a working method through tool beta testing (WG3WP4 D7)
- The development of a protocol for interfacing with an existing shallow water solver (WG3WP4 D10)

7 REFERENCES

1 International Electrotechnical Commission (2010) PT 62600-3-2 TS (Draft). *Tidal energy resource characterisation and assessment*

2 ABP Marine Environmental Research Ltd (2004) *Atlas of UK Marine Renewable Energy Resources: Technical Report*. Department of Trade and Industry. Report number: R.1106

3 Chanson, H. (2004) *The Hydraulics of Open Channel Flow: An Introduction*. 2nd edition. Oxford, Elsevier Butterworth-Heinemann

4 Hervouet, J.-M. (2007) *Hydrodynamics of Free Surface Flows: Modelling with the Finite Element Method*. Chichester, John Wiley & Sons Ltd

5 Wikipedia (no date) *Shallow water equations*. [Online] Available from: http://en.wikipedia.org/wiki/Shallow_water_equations

6 TELEMAC (no date) *The TELEMAC modelling system*. [Online] Available from: http://www.telemacsystem.com/index.php?option=com_content&view=article&id=5&Itemid=2&lang=en

7 HR Wallingford Ltd & Departement Laboratoire National d'Hydraulique Electricité de France (no date) *TELEMAC*. [Online] Available from: http://kenai.com/downloads/telemac/brochures/Telemac.pdf

8 Pham, C.-T. & Martin, V. A. (2009) *Tidal current turbine demonstration farm in Paimpol-Bréhat (Brittany): tidal characterisation and energy yield evaluation with Telemac*. The 8th European Wave and Tidal Energy Conference (EWTEC 2009), Uppsala, Sweden

9 Cornett, A., Durand, N., Serrer, M. (2010) *3-D Modelling and Assessment of Tidal Current Resources in the Bay of Fundy, Canada*. The 3rd International Conference on Ocean Energy (ICOE 2010), Bilbao, Spain

10 Amoudry, L. *et al* (2009) *A Scoping Study on: Research into Changes in Sediment Dynamics Linked to Marine Renewable Energy Installations*

11 TELEMAC (no date) *TELEMAC System Software*. [Online] Available from: http://www.telemacsystem.com/images/licence/Licence_Telemac.pdf

12 DHI (no date) *Coast and sea*. [Online] Available from: http://www.mikebydhi.com/Products/CoastAndSea.aspx

13 Lawrence, J., Kofoed-Hansen, H., Chevalier, C. (2009) High-resolution metocean modelling at *EMEC's (UK) marine energy test sites*. The 8th European Wave and Tidal Energy Conference (EWTEC 2009), Uppsala, Sweden

14 ANSYS (no date) *ANSYS CFD*. [Online] Available from: http://www.ansys.com/assets/brochures/ansys-cfd-brochure.pdf

15 Versteeg, H. K. & Malalasekera, W. (2007) *An introduction to computational fluid dynamics: the finite volume method*. 2nd edition. Harlow, Pearson Education Ltd

16 CD-adapco (no date) *STAR-CD V4: From Fluid Dynamics to Continuum Mechanics*. [Online] Available from: http://www.cd-adapco.com/products/brochures/STAR-CD/STAR-CD.pdf

17 Gerris (no date) *Gerris Flow Solver*. [Online] Available from: http://gfs.sourceforge.net/wiki/index.php

18 Office of Naval Research (no date) *ROMS (Regional Ocean Model System)*. [Online] Available from: http://www.ocean-modeling.org/index.php?page=models&model=ROMS

19 IMCS: Ocean Modeling Group (no date) *Regional Ocean Modeling System (ROMS)*. [Online] Available from: http://www.myroms.org/

20 Delft Hydraulics (no date) *Delft3D suite*. [Online] Available from: http://delftsoftware.wldelft.nl/index.php?option=com_content&task=blogcategory&id=13&Itemid=34

21 ADCIRC (no date) *ADCIRC Coastal Circulation and Storm Surge Model*. [Online] Available from: http://www.unc.edu/ims/adcirc/

22 AQUAVEO (no date) *ADCIRC Model*. [Online] Available from: http://www.aquaveo.com/html/sms/models/adcirc.html

23 Trigo-Teixeira, A. & Rebordão, I. (2009) *An assessment of tidal energy potential. The Lima estuary*. The 8th European Wave and Tidal Energy Conference (EWTEC 2009), Uppsala, Sweden

24 SMIC (no date) *RMA10*. [Online] Available from: http://smig.usgs.gov/cgi-bin/SMIC/model_home_pages/model_home?selection=rma10

25 International Electrotechnical Commission (2010) PT 62600-200 TS (Draft). *Power performance assessment for electricity producing tidal energy converters*

26 (Adapted from): The Engineering Toolbox (no date) *Hydraulic diameter*. [Online] Available from: http://www.engineeringtoolbox.com/hydraulic-equivalent-diameter-d_458.html

27 Whelan, J. I., Graham, J. M. R., Peiró, J., (2009) *A free surface and blockage correction for tidal turbines*. Journal of Fluid Mechanics 624, 281-291.

28 Randall, D. (2006) *The Shallow Water Equations*. [Online] Available from: http://kiwi.atmos.colostate.edu/group/dave/pdf/ShallowWater.pdf

Appendix 1: Derivation of the shallow water equations

(See separate document[28])