Internship Report

Implementation of Tidal Energy Extraction in a Finite Volume Coastal Ocean Model

Rob Hiley

ETP PhD Student, Heriot-Watt University, Orkney Campus

in partnership with

Marine Scotland Science, Marine Laboratory, Aberdeen
This internship was funded by NERC under the MRE Knowledge Exchange Programme, and hosted by the Oceanography Group, Marine Scotland Science, at the Marine Laboratory, Torry, Aberdeen.

Special thanks are due to my supervisors, Rory O’Hara Murray and Alejandro Gallego at MSS, and Rob Harris at Heriot-Watt University, International Centre for Island Technology (ICIT), for their constant support and encouragement. I should like to thank the programme coordinators, Annie Linley and Eleanor Ashton, for giving me this opportunity. Thanks also to many colleagues at ICIT and the Marine Laboratory for helpful discussions and friendly support, in particular, to Inga Burton, Susana Bastón and Simon Waldman at ICIT, and Gustavo Sanchez, on placement at the Marine Laboratory, from the University of Aberdeen.
# Contents

1 Introduction 4

2 Background 5

3 Aims and Approach 8

4 Physical Effects 14
   4.1 Drag ........................................ 14

5 Flow Modelling 16
   5.1 Continuum Mechanics ....................... 16
   5.2 Momentum Balance .......................... 18
   5.3 Free-Surface Flow .......................... 19

6 Energy Extraction Modelling 23
   6.1 Momentum Sink ............................. 23
   6.2 Actuator Disk ............................... 24
   6.3 Bed Stress ................................. 25
   6.4 Secondary Effects .......................... 27

7 Finite Volume Method 27
   7.1 Spatial Discretization ........................ 27
   7.2 Discrete Balance .......................... 28
   7.3 Numerical Solution ....................... 30

8 Implementation 31

9 Idealized Channel Simulation 32
   9.1 One-dimensional Model ..................... 36

10 Conclusions 39

A One-Dimensional Modelling 45
1 Introduction

The technology to exploit the power of the tides in coastal waters has reached the point where large-scale deployment of marine renewable energy (MRE) devices is a realistic prospect. There is a current need to obtain detailed estimates of the power that might be delivered by arrays of these devices at specific sites, and of any potentially significant environmental effects. The only tools really suited to this purpose are computational ocean dynamics models. These calculate the detailed time-varying flow field by solving the discretized equations of motion, subject to specified boundary conditions.

Whilst the use of computational models for the calculation of ocean flows is well established, their application to the modelling of MRE installations is a recent development, requiring new techniques. Initially, a device can be represented as an area of locally increased bed roughness; this allows existing models to be used without modification. Ultimately, existing models may have to be modified or extended to accommodate the range of anticipated effects.

This report describes progress in a recent internship focused on developing the means to represent tidal stream energy extraction in the Finite Volume Community Ocean Model FVCOM (Chen et al., 2013). The work is part of a wider effort to develop a computational ocean modelling tool capable of assessing the performance and hydrodynamic effects of proposed MRE installations at local and regional scales.

Direct validation of ocean models is possible to a very limited extent. Usually, few data are available for comparison; measurements are limited to averaged values of key variables; and the boundary data required to force the model are highly uncertain. Confidence in the quality and suitability of a model is based instead on the soundness of the theory and the quality of the software. Confidence in the output depends also on the accuracy of the solution and the skills of the user. There is therefore a need for a theoretical framework within which the principles underlying the formulation of these models and the interpretation of their results can be clearly understood.
The main objective of this internship was to provide a flexible software module with the potential to accommodate a wide range of device types, to serve as a platform for future model development. The initial implementation is restricted to the representation of drag. More sophisticated models, taking account of secondary effects such as the production of small-scale turbulence, may be implemented in the future. The theoretical framework, presented here in outline, and in more detail in the project technical report, is designed to accommodate these future developments.

An idealized channel model was set up as a demonstration test-case. The original intention was to carry out a detailed comparison with the results obtained from a one-dimensional model. Difficulties associated with the representation of boundary conditions prevented a rigorous comparison in the time available. The study has nevertheless identified the potential value of one-dimensional models as a means of partial validation, and has provided a framework within which this might be explored.

2 Background

MRE resources. Scotland has substantial tidal energy resources, and the Scottish Government is actively encouraging the development of the technology and infrastructure to take advantage of this endlessly renewable, virtually pollution-free, and entirely predictable resource. The Pentland Firth (figure 1), which separates Orkney from mainland Scotland, has been a focus of interest for the exploitation of tidal stream energy, because of the high flow speeds generated by the difference in tidal phase between the Atlantic Ocean and the North Sea (Easton et al., 2012).

Despite the apparent environmental advantages of this energy technology, there is concern to ensure that any adverse effects are acceptable, and research is being pursued worldwide that will lead to a better understanding of the potential environmental and ecological consequences of MRE developments.
Modelling and assessment. Marine Scotland is a directorate of the Scottish Government with overall responsibility for the integrated management of Scotland’s seas. Marine Scotland Science (MSS), as part of that directorate, is concerned directly with scientific research and assessment. One role of the Oceanography Group within MSS is to advise on the potential effects of any proposed MRE developments on the physical environment. This advice is based mostly on the examination of model output supplied by developers, and qualitative assessments based on conceptual understanding of oceanographic and coastal processes. There is a need for a more comprehensive modelling capability by which (i) to obtain more detailed and reliable estimates of available energy resources, (ii) to quantify potential interactions between multiple installations, and (iii) to assess combined, cumulative effects on a range of environmentally and ecologically significant processes.

Simple one-dimensional models can be used to obtain estimates
of the maximum energy resource in a tidal channel. Detailed estimation of the actual resource, and quantification of the hydrodynamic effects, however, requires more sophisticated computational ocean dynamics models. MSS has adopted FVCOM (Chen et al., 2013) as its preferred modelling system for application to the coastal and shelf seas of Scotland, and has commissioned a number of specific model configurations, including one of the whole Scottish Shelf, and a high-resolution model of the Pentland Firth and Orkney Waters. FVCOM is an open-source finite volume ocean modelling system, with an active community of developers and users. At present it has no built-in facility for the representation of tidal energy extraction.

**Ocean modelling.** Coastal ocean flows have a number of distinguishing characteristics which set them apart from other flows. These include: a dynamically varying free surface; complex bottom terrain; irregular, time-varying coastal boundaries; a wide range of interacting scales of motion; and variable fluid mass density.¹ Also, the simulation of ocean flows in limited regional areas necessitates the use of open boundaries. Apart from the problem of missing or uncertain data, open boundary conditions must be designed to allow free propagation of outgoing waves and dissipation of computational noise (see Herzfeld et al., 2011).

Computational ocean models have evolved somewhat separately from other computational fluid dynamics (CFD) models. A variety of numerical methods have been developed to solve the governing equations of oceanic motion, with a range of techniques to enhance the stability, accuracy, or efficiency of the solution procedure. The use of these models requires expertise, in setting up the problem, and tuning the solution parameters. In interpreting the results, users must rely on their own understanding of, not only the behaviour of ocean systems and the underlying physics, but also the numerical procedures by which the governing equations are solved. To support this work, there is a need for a clearly structured statement of the

¹This may be important in sea lochs and estuaries.
fundamental principles upon which these models are constructed. The relevant principles are those of physics, mathematics, and software engineering.\textsuperscript{2}

\textbf{MRE modelling.} Conventional CFD models have been applied to the study of individual devices under laboratory conditions (e.g. Sun et al., 2008); and high-resolution results have been obtained for realistic scenarios, subject to the simplifying assumption of steady flow conditions (Crammond et al., 2013). Usually though, tidal flow modelling studies are constrained by computational resources, and the mesh is comparatively coarse. To be useful for validation purposes, the results of any such detailed studies have to be interpreted in terms of overall averages, and care must be taken to avoid mesh dependence in the problem formulation.

\section{Aims and Approach}

The objective of the internship was to provide a self-contained, user-programmable module, (i) enabling the user to implement a variety of device-specific performance curves, and (ii) serving as a platform for the development of more sophisticated models.

Given the arguments above, it is important to establish a clear theoretical framework that can serve as a basis for (i) the consistent formulation and inter-comparison of models; (ii) the comparative assessment of different approximations and solution techniques; (iii) the calibration of coarse or averaged models using the results of more detailed modelling studies; and (iv) the verification of complex models against simpler model solutions for idealized test cases. An underlying aim of the project, therefore, was to clarify the theoretical framework relating to ocean dynamics modelling in general, and the representation of tidal stream energy extraction in particular.

\textsuperscript{2}Mathematics as a language, not just the science of arithmetic. The construction, or specification, of a mathematical theory is in many ways similar to that of a software system.
What are the effects of MRE devices on the tidal dynamics?

What is the nature of the flow?

How is the flow modelled?

Coastal ocean dynamics
- wide range of scales of motion
- dynamic free surface
- complex bathymetry and coastline
- meteorological forcing

Continuum mechanics
- primitive ocean equations
- open boundaries
- external and internal modes
- turbulence model

Differential balance equation
\[ \Psi + \nabla \cdot \Phi = \Sigma \]

Integral balance equation
\[ \int_V \Psi dV + \int_{\partial V} \Phi \cdot n dA = \int_V \Sigma dV \]

Storage fluxes sources

Local effects
- drag; thrust
- swirl; vorticity
- small-scale turbulence
- dissipation of tidal eddies

Wake effects
- device interactions
- kinetic energy dissipation
- shear-induced turbulence

MRE device model
- discontinuous flux or distributed source
- empirical drag law

Finite volume method
- mode-split time-stepping
- unstructured mesh

Software implementation
- modular design

Mathematical foundations
- Tensor theory; differentiation
- Measure theory; integration
- Geometry: kinematics
- Coordinate transformation

What confidence do we have in the results?
- Is the model valid?
- Is the software correct?

Figure 2: Flow diagram. The colours indicate different areas of study: oceanography, fluid dynamics, mathematics and physics of continua, numerical methods and software engineering.
The question of how best to simulate the hydrodynamic effects of MRE devices leads to a number of more specific questions relating to various aspects of the problem, as illustrated in fig. 2. Preliminary considerations relating to some fundamental elements of the theory and the model implementation are discussed below, and developed further in subsequent sections.

**Continuum physics.** The continuum balance equations are the starting point for the analysis of any flow or deformation problem. They express fundamental principles governing the rates of change of physical properties such as mass and energy. Subject to the specification of appropriate constitutive relations, they completely determine the motion of any idealized material body.

The set of balance and constitutive equations together form the *equations of motion*. Two- and one-dimensional forms of these equations can be derived by integration. Since they are non-linear, this gives rise to additional terms, similar to Reynolds stresses, which cannot be expressed as functions of the averaged state variables.

The same process of integration, repeated, leads to the semi-discretized equations of motion, which form the basis of the finite volume method. The spatial integration procedure transforms the *partial* differential equations of motion—whose solution comprises a set of maps between continua—into a set of coupled *ordinary* differential equations involving a finite number of discrete variables. These are less precise than the continuum equations, but they give an exact spatially-discretized description of an idealized continuous medium, and can be solved approximately by numerical methods.

The conventional interpretation of the Reynolds stress tensor is as a description of turbulence. More generally, it is simply a measure of the sub-integral scale motion, which may be turbulent or not. It is a direct consequence of advective transport, and the effect is not restricted to momentum: a similar term arises in the balance equation of every advected quantity other than volume.
One-dimensional modelling. The one-dimensional channel flow equations, in the form of the Saint-Venant equations (Sturm, 2001, §7), are well understood by civil engineers. Given the impossibility of direct validation, this presents an alternative prospect, whereby a complex three-dimensional model might be partially validated against a trusted one-dimensional model of known accuracy.

It was originally proposed that an idealized channel problem be used in the project as a demonstration test-case for the modelling of MRE extraction. Given the argument above, and consistent with the mathematical aims of the project, the specific objective then was to derive the channel flow equations in exact form, as one-dimensional balance equations. It became apparent, as the work progressed, that this result is obtained naturally from the depth-integrated balance equation, by applying the same integration procedure, this time integrating in the cross-channel direction. In fact, the same procedure gives, precisely: (i) the channel-flow equations, in continuous and semi-discrete form; (ii) the layer-averaged balance equations, similarly; and (iii) the control-volume averaged balance equations, which constitute the basis of the finite volume method.

Given that the general balance equation expresses, in essence, all of the transport equations of continuum physics, this appears to be a very useful and worthwhile result. It provides a strong theoretical foundation with very wide applicability. From the point of view of the current project, it gives some insight into the general questions of how to interpret model results—as integral-scale results—and how to model sub-integral scale motion.

Scales of motion. A choice of scale(s) is implicit in the formulation of a continuum model. In principle, the differential equations of motion describe the behaviour of a fluid at every scale; in practice, any real flow measurement, and any value computed by a model, is associated with some finite region of space and time. In the finite volume method, the resolved length scale is determined implicitly by the mesh: the flow variables are formally defined as averages taken
over a prescribed set of control volumes. This spatial discretization can be interpreted as the result of an implicit filtering operation (Rogallo and Moin, 1984).³

**Tensor language.** The continuum equations are naturally and concisely expressed in the language of tensors. In mathematics texts, tensor products are defined in purely algebraic terms, as multilinear maps between vector spaces (Dodson and Poston, 1991, §V). Tensor product spaces are linear (i.e. vector) spaces, of higher rank than those on which they operate.⁴ Thus, they are subject to the usual rules of linear algebra, and all of the results of vector space theory apply automatically to tensors. The derivative of a map, when it exists, is defined as a linear map between (tangent) vector spaces. Thus, the tensor theory provides the foundation upon which the theory of differential calculus is formally constructed.

In physics texts, on the other hand, vectors and tensors are usually introduced as collections of components which transform according to certain rules (e.g. Aris, 1962, §2), and attention is often restricted to Euclidean spaces, in order to simplify the algebra. The inner product, or metric tensor, which is responsible for the geometry of the space, may then be eliminated from consideration. When non-orthogonal coordinate systems are involved, however, the algebraic rules become more complex.

An attempt has been made in this project, to present the general tensor theory in a readily accessible and transparent form. A self-contained account is given in the technical report produced for the project. One advantage of the direct tensor notation is that the governing equations are expressed in a form that does not depend on the choice of coordinate system, whereas any specific coordinate-based form may be derived by straightforward application of alge-

³Filtering is the basis of the approach to turbulence modelling known as *large eddy simulation* (Germano, 1992). In measure theory (e.g. Krantz and Parks, 2008) a filter may be thought of as a probability distribution function. A volume integral is equivalent to a simple ‘box’ filter (see Berselli et al., 2006, §1).

⁴Tensor algebra can be viewed as a generalization of matrix algebra.
Software design. In principle, the algebraic tensor notation can be translated directly into program code, with tensors as abstract data structures. Data abstraction and encapsulation are important concepts in the modular approach to software design. Application of the principles of modular design can lead to dramatic improvements in the robustness, clarity, and re-usability of program code. The proposed implementation of the energy extraction model takes the form of a Fortran 95 module with an explicitly defined interface. This greatly simplifies the task of code verification by inspection, and demarcates the boundary within which new developments may be implemented safely, without risk of side-effects. These software design considerations are important for several reasons, not least that the aim of the project is to provide a platform for development.

Modules. Fortran 95 modules may contain variable declarations, data structure definitions, and module procedures (i.e. functions or subroutines) (Adams et al., 1997, §11). Good practice dictates that the contents of a module should be declared PRIVATE by default; then the only entities that are available to other program units are those explicitly named in a PUBLIC statement. The totality of those entities constitutes the module’s public interface; private entities are not accessible to any other program unit. The modifications required to accommodate a new module in an existing program depend only on the module’s public interface. This provides the ideal environment to experiment with new models.

---

5 This gives “a physical insight into the phenomena under discussion which are, in general, made unnecessarily obscure by expression in particular coordinate systems. The proper function of coordinates is to perform the final step of algebraic interpretation.” (Milne-Thomson, 1960, Preface)

6 The essential principle is that the complexity of any large problem can be reduced by means of modular decomposition (see Parnas, 1972).
4 Physical Effects

The primary hydrodynamic effect of an MRE device is the drag force due to the pressure difference that develops between its upstream and downstream facing surfaces. This provides the motive force from which useful power is obtained. The direct effect on the flow is a loss of momentum. In general, there are also a number of secondary effects. (i) With axial-flow turbines, some of the axial momentum of the free stream is converted to vorticity, which is transported downstream in the form of a swirling wake. (ii) Devices operating in shallow water, or close to the surface, are subject to fluctuating loads due to the action of surface waves (see Sarpkaya and Isaacson, 1981). (iii) Locally generated eddies may lead to the production of turbulence at small scales.

Tidal flows in general contain transient flow structures at a wide range of scales; and large tidal eddies may cause significant transients in the loadings on MRE devices. Background turbulence may also affect device performance, and is known to influence wake development (Mycek et al., 2014). If the modelling objective is to obtain accurate estimates of the performance of a specific installation, these effects should be taken into account.

4.1 Drag

Any obstacle placed in a flow experiences a drag force, exerted by the fluid, in the direction of flow. The fluid is subject to an equal and opposite retarding force. The nett drag force on a bluff (as opposed to streamlined) body consists of two parts: (i) skin friction, due to the action of viscous shear forces that arise in consequence of the fluid-solid boundary conditions; and (ii) form drag, due to the unevenly distributed normal pressure forces acting on the surfaces of the body.

---

7 "Wave loads are one of the main contributors to fatigue loads of tidal turbine blades." (Faudot and Dahlhaug, 2012)
A characteristic Reynolds number may be defined as $R = uL/\nu$, where $u$ is the (notional) free stream velocity, $\nu$ is the kinematic viscosity of the fluid, and $L$ is some length-scale characterizing the device. As $R$ increases, the significance of skin friction reduces. The viscous forces nevertheless govern the development of the boundary layers, which determine the size of the wake, and this has a significant effect on form drag. Viscous forces also contribute to momentum transfer at the boundary of the wake; and this may lead to instability and further turbulent mixing. Note that the onset of turbulence in an initially laminar boundary layer can lead to a sudden reduction in the drag force on a bluff body, due to the delayed separation of the boundary layer, resulting in a narrower wake, and consequently, reduced form drag (Tritton, 1977, §3).

The nett drag force $D$ on an obstacle may be expressed in terms of a dimensionless drag coefficient $C_D$ (Tritton, 1977, §7):

$$C_D = \frac{D}{\frac{1}{2} \rho u^2 L^2},$$

where $\rho$ is the fluid mass density. Dimensional analysis suggests that $C_D$ may be a function of the Reynolds number $R$, and other dimensionless ratios determined by the geometry of the flow.

Experimental observations of the drag forces on simple shapes (e.g. cylinders) show that, at high Reynolds numbers, $C_D$ attains an approximately constant value for a given geometry (under steady inflow conditions). This is explained by Bernoulli’s equation for an inviscid fluid in steady irrotational flow (Tritton, 1977, §10). Under these conditions, the stagnation pressure $p + \frac{1}{2} \rho u^2$ remains constant along streamlines. It follows that the drag force, which in the inviscid limit is due to form drag, is approximately proportional to $u^2$.

---

8The drag or skin-friction coefficient for wall-bounded shear flow is sometimes defined without the factor $\frac{1}{2}$ in the denominator (see §6.3).

9“[T]he drag and inertia coefficients are not time-invariant and depend on the Reynolds number, relative motion of the fluid, history of the motion, relative roughness, etc.” (Sarpkaya and Isaacson, 1981, §2.8). More generally, the drag coefficient may be defined as a tensor (Mattis et al., 2012).
At (very) low Reynolds numbers, on the other hand, experiments suggest that $C_D$ is inversely proportional to the Reynolds number $\mathcal{R}$. In this case, the drag force $D$, due mostly to skin friction, is directly proportional to the flow velocity $u$.

The actual value of $C_D$ that is used in a simulation, and the functional form of its dependence on $\mathcal{R}$ (and any other dimensionless parameters) should be chosen in accordance with the characteristics of the device in question, based on the results of experiment (Bahaj et al., 2007), or detailed simulation (Batten et al., 2008), or ideally both (Yang and Lawn, 2011).

5 Flow Modelling

5.1 Continuum Mechanics

The thermomechanical state of motion of any idealized material body is defined mathematically by certain of its extensive properties, whose rates of change are governed by physical balance laws (see Ziegler, 1983). In ocean modelling, the relevant properties are mass, linear and angular momenta, energy, entropy, and salt content (Müller, 2006, §3) (see also Shapiro, 1961).

A material body is defined as a set of notional particles whose motion is described by the material velocity field $\mathbf{v}(\mathbf{x}, t)$, where $\mathbf{x}$ denotes a location in space, and $t$ is an instant in time. Consider a material body $\mathcal{B}$ whose instantaneous configuration is $\Omega(t)$.

General balance. Let $\psi(\mathbf{x}, t)$ be any tensor field representing a density—i.e. an amount per unit volume—of some extensive thermomechanical property. The total amount of the property $\psi$ con-

---

10 The material velocity field may be chosen arbitrarily. Considering sea water as a two-component mixture of water and dissociated salt, each with its own independent velocity field, we choose the barycentric (i.e. density-weighted average) velocity (Müller, 2006, §3); transport of total mass is then purely advective.

11 The configuration is the set of locations occupied by the particles of a body.

12 Vectors and scalars are tensors of rank one and zero respectively.
tained within the body $B$ is given by the volume-integral $\int_{\Omega(t)} \psi \, dV$. The balance law for $\psi$ simply states that the rate of change of this total amount is balanced by the nett influx of $\psi$ across the boundary $\partial\Omega(t)$ of the body, plus the nett rate of supply or production of $\psi$ due to external or internal sources. We suppose that the body $B$ may contain (part of) a singular surface $S$, which may contribute actively to the balance of $\psi$. The general balance equation (Liu, 2002, §2) may then be written:

$$\frac{d}{dt} \int_{\Omega(t)} \psi \, dV + \int_{\partial\Omega(t)} \mathbf{n} \cdot \varphi \, dA = \int_{\Omega(t)} \varsigma \, dV + \int_{S(t) \cap \Omega(t)} \varsigma_S \, dA,$$

where: $\varphi(x,t)$ is the flux tensor field; $\varsigma(x,t)$ is the source density field; $\varsigma_S(x,t)$ is the surface density of production on $S(t)$ (Hutter and Jöhnk, 2004, §3); $\mathbf{n}(x,t)$ is the unit outward-normal dual vector field defined on $\partial\Omega(t)$; and $A$ is the area measure. Balance equations in integral form admit discontinuous solutions, and can be used to model hydraulic jumps, flood waves, and other discontinuous phenomena (Dick, 1989, §1).

**Field equation.** If the field variables $\psi$ and $\varphi$ are continuously differentiable on $\Omega(t)$ (which implies $S(t) \cap \Omega(t) = \emptyset$), and the boundary $\partial\Omega(t)$ is piecewise smooth, then the left-hand side of equa-

---

13 "An oriented smooth surface $S$ in a material region $V$ is called a singular surface relative to a field $\psi$ defined on $V$, if $\psi$ is smooth in $V - S$ and suffers a jump discontinuity across $S$. The jump of $\psi$ is defined as

$$\jump{\psi} = \psi^+ - \psi^-,$$

where $\psi^+$ and $\psi^-$ are the one-side limits from the two regions of $V$ separated by $S." (Liu, 2002, §2.1)

14 Liu (2002) does not include surface production in the derivation of the general balance equation. Hutter and Jöhnk (2004) take the surface production to be zero except in the entropy balance equation. Our motivation is to show how actuator disks (see §6.2) are accommodated by the continuum theory.

15 The ranks of $\varsigma$ and $\varsigma_S$ are equal to that of $\psi$; the rank of $\varphi$ is one higher. The flux tensor is defined such that $\mathbf{n} \cdot \varphi$ is an outflux density.

17
tion (2) may be written as a volume-integral, using Reynolds’ transport theorem and Green’s theorem (Aris, 1962, §3). This gives:

\[
\int_{\Omega(t)} \left( \frac{\partial \psi}{\partial t} + \nabla \cdot (\mathbf{v} \otimes \psi + \boldsymbol{\varphi}) - \varsigma \right) dV = 0. \tag{3}
\]

Since the domain of integration is arbitrary, the integrand itself must be zero; thus, the following field equation holds at regular points:

\[
\frac{\partial \psi}{\partial t} + \nabla \cdot (\mathbf{v} \otimes \psi + \boldsymbol{\varphi}) = \varsigma. \tag{4}
\]

**Jump condition.** Suppose \( \Omega(t) \) contains a singular surface (i.e. \( S(t) \cap \Omega(t) \neq \emptyset \)). The balance equation (2) in this case implies a jump condition, which holds at singular points:

\[
n_S \cdot \left[ (\mathbf{v} - \mathbf{v}_S) \otimes \psi + \boldsymbol{\varphi} \right] = \varsigma_S, \tag{5}
\]

where \( n_S(x, t) \) is the unit dual vector field normal to \( S(t) \), and \( \mathbf{v}_S(x, t) \) is the displacement velocity of \( S(t) \). If \( S \) is fixed (\( \mathbf{v}_S = 0 \)) and has no sources of production (\( \varsigma_S = 0 \)), then the jump condition (5) states that the total flux \( n_S \cdot (\mathbf{v} \otimes \psi + \boldsymbol{\varphi}) \) must be continuous across \( S \).

The above equations apply to the transport of any quantity that can be regarded as a density whose rate of change is balanced by fluxes and sources.

### 5.2 Momentum Balance

Balance of linear momentum is expressed by Cauchy’s equation (Aris, 1962, §5), which applies to any idealized continuous medium:

\[
\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \otimes \mathbf{v} - \mathbf{\sigma}) = \rho \mathbf{f}, \tag{6}
\]

\(^{16}\nabla \cdot \varphi = \text{tr}(\nabla \varphi)\) denotes the divergence of a tensor field \( \varphi \), where \( \text{tr} \) is the trace operator, and \( \nabla \varphi \) denotes the gradient, whose components are the covariant derivatives of \( \varphi \) (Liu, 2002, §A).

\(^{17}\)A regular point is one at which the field variables vary smoothly.
where $\rho(x, t)$ is the mass density, $\sigma(x, t)$ the stress tensor, and $f(x, t)$ the specific body force vector field. This has the form of the general balance equation (4), with $\psi = \rho \mathbf{v}$, the linear momentum density, $\varphi = -\sigma$, the non-advective momentum flux tensor,\(^{18}\) and $\zeta = \rho f$, the body force density.

For fluids, the stress tensor $\sigma$ is a sum of two independent terms:

$$\sigma = -p \bar{g} + \tau, \tag{7}$$

where $p(x, t)$ is the thermodynamic pressure, $\bar{g}(x, t)$ is the dual inner product (or conjugate metric tensor),\(^ {19}\) and $\tau(x, t)$ is the viscous stress tensor (Aris, 1962, §8). Isotropic, incompressible, Newtonian fluids are characterized by $\tau = \mu \mathbf{d}$, where $\mu$ is the dynamic viscosity of the fluid, and $\mathbf{d}$ is the strain-rate tensor (i.e. the symmetric part of $\nabla \mathbf{v}$). The incompressible Navier-Stokes equations are obtained by supposing the fluid mass density $\rho$ and dynamic viscosity $\mu$ (or kinematic viscosity $\nu = \mu / \rho$) to be material constants.

For geophysical flows, the specific body force vector $f$ is given by

$$f = g - 2\omega \times \mathbf{v}, \tag{8}$$

where $g$ is the gravitational acceleration vector, and $\omega$ is Earth’s angular velocity vector (Müller, 2006, §3).

### 5.3 Free-Surface Flow

In ocean modelling, the flow domain is bounded above by the free surface, and below by the bed. In general, these bounding surfaces are neither fixed nor material. The free surface is dynamic, and subject to mass transfer due to precipitation and evaporation. The bed geometry may change due to erosion and deposition of sediments; and there may be mass transfer due to groundwater exchanges.

\(^{18}\)The term ‘advective’ always implies a particular choice of ‘material’ velocity field. Any transport not associated with that velocity field is ‘non-advective’.

\(^{19}\)In a Euclidean space $\bar{g}$ may be identified with the identity tensor.
In this section, an arbitrary location in \( n \)-dimensional space is denoted by \((x, z)\), where \( x \) and \( z \) represent horizontal and vertical locations respectively. It is assumed that the natural basis vectors of the coordinate system do not vary with \( z \).\(^{20}\) Let \( \zeta(x, t) \) and \( b(x, t) \) denote the elevations (i.e. \( z \)-coordinates) of the free surface and the bed respectively.\(^{21}\) The free surface is conveniently described as the zero-contour of the scalar field \( Z(x, z, t) = z - \zeta(x, t) \). Similarly, the bed is the zero-contour of \( B(x, z, t) = z - b(x, t) \). The depth of water is defined (for any \( z \)) as

\[
H(x, t) = \zeta(x, t) - b(x, t) = B(x, z, t) - Z(x, z, t).
\]

**Depth-average.** The depth-average of a field \( \psi \) is defined as

\[
\overline{\psi}(x, t) = \frac{1}{H(x, t)} \int_{\zeta(x, t)}^{\zeta(x, t)} \psi(x, z, t) \, dz.
\]

The balance equation (4) for \( \psi \) may be integrated with respect to the vertical coordinate \( z \), to obtain an \((n - 1)\)-dimensional\(^{22}\) field equation for \( H \overline{\psi} \):

\[
\frac{\partial H \overline{\psi}}{\partial t} + \nabla \cdot \left( H \overline{v} \otimes \overline{\psi} + H \overline{\varphi} \right) = H \zeta - \varsigma_Z + \varsigma_B, \tag{9}
\]

where \( \varsigma_Z(x, t) \) and \( \varsigma_B(x, t) \) represent the flux densities of \( \psi \) at the top and bottom surfaces, per unit plan area. These are given by:\(^{23}\)

\[
\varsigma_Z = V_Z \psi \zeta + (\nabla Z) \cdot \varphi \zeta, \quad \varsigma_B = V_B \psi b + (\nabla B) \cdot \varphi_b, \tag{10}
\]

\(^{20}\)The natural basis of a coordinate system is a field of bases whose vectors are tangent to the coordinate curves (Liu, 2002, §A). If these basis vectors depend on \( z \) then their gradients contribute to the depth-integral of the divergence of an arbitrary tensor field.

\(^{21}\)Since \( \zeta \) and \( b \) are single-valued there can be no breaking waves or overhangs.

\(^{22}\)This refers to the dimension of the space of locations. The tensor character of \( \overline{\psi} \) is the same as \( \psi \).

\(^{23}\)In practice, the boundary conditions are imposed externally; the boundary may be regarded as a singular surface.
where \( V_Z(x, t) \) and \( V_B(x, t) \) are the volume flux densities, \( \nabla Z \) and \( \nabla B \) are surface-normal dual vectors, and

\[
\psi_\zeta(x, t) = \psi(x, \zeta(x, t), t) \quad \text{and} \quad \psi_b(x, t) = \psi(x, b(x, t), t)
\]

are the values of \( \psi \) at the top and bottom surfaces respectively. The surface flux tensors \( \varphi_\zeta \) and \( \varphi_b \) are defined similarly. Equation (10) generalizes the usual kinematic boundary conditions, in which the top and bottom boundaries are regarded as material surfaces, i.e. \( V_Z = V_B = 0 \).

Note that the source terms \( \varsigma_Z \) and \( \varsigma_B \) represent external fluxes at the boundaries of the \( n \)-dimensional domain; consequently, they do not appear in the field equation (4). In the \((n-1)\)-dimensional domain of horizontal locations they have become internal sources.

**Effective flux.** Let \( \psi' = \psi - \bar{\psi} \) denote the deviation of a field \( \psi \). Equation (9) may be cast in the form of the general field equation (4) by writing

\[
\mathbf{v} \otimes \psi + \varphi = \mathbf{v} \otimes \bar{\psi} + \varphi^*, \quad \text{where} \quad \varphi^* = \varphi + \mathbf{v}' \otimes \psi'
\]

is an ‘effective flux’ tensor. Thus, the averaging process gives rise to an additional ‘deviation flux’, represented by the tensor \( \mathbf{v}' \otimes \psi' \), which is a measure of the correlation between the vertical deviation profiles of \( \mathbf{v} \) and \( \psi \).\(^{24}\) In the case of linear momentum \( (\psi = \rho \mathbf{v}) \), the deviation flux has the form of the Reynolds stress tensor.\(^{25}\)

**Volume balance.** The \( n \)-dimensional volume balance, or continuity equation may be written

\[
\nabla \cdot \mathbf{v} = \dot{\varepsilon},
\]

\(^{24}\)This additional flux is advective (in \( n \) dimensions) with respect to \( \mathbf{v} \), but non-advective with respect to (i.e. not transported by) \( \nabla \).

\(^{25}\)Reynolds stresses are traditionally defined as temporal or ensemble averages and interpreted as measures of turbulence (e.g. Kay and Nedderman, 1985, §6).
where $\dot{\varepsilon}(x, z, t)$ is the volumetric strain rate. Flows for which $\dot{\varepsilon}$ is identically zero are called isochoric. This effectively means that the mass density of each fluid particle is constant with respect to time, which is a reasonable approximation for most coastal ocean flows. In general, $\dot{\varepsilon}$ is related to changes in other state variables, notably temperature and salinity, via the thermodynamic equations of state.

The volume balance equation (12) has the form of the general balance equation (4), with $\psi = 1$, $\varphi = 0$, and $\varsigma = \dot{\varepsilon}$. The depth-integrated balance equation (9) in this case is

$$\frac{\partial H}{\partial t} + \nabla \cdot (H \mathbf{v}) = H \dot{\varepsilon} - V_Z + V_B. \tag{13}$$

An evolution equation for $\overline{\psi}$ (as opposed to $H \overline{\psi}$) is obtained by multiplying equation (13) by $\overline{\psi}$ and subtracting from equation (9).

**Hydrostatic pressure.** If the flow velocity $\mathbf{v}$ is (momentarily) zero (which implies $\tau = 0$), substitution of equations (7) and (8) into the momentum balance equation (6) gives $\nabla p = \rho g$; and the vertical component of the depth-average balance equation (9) gives $p_b = p_\zeta + gH \overline{\rho}$, where $g = |g|$ is the gravitational constant. Similarly, the hydrostatic pressure $p_h$ is defined, for any vertical location $z$, according to the static balance of vertical forces:

$$p_h(x, z, t) = p_\zeta(x, t) + \int_z^\zeta(x, t) \rho(x, z', t) \, dz'. \tag{14}$$

**$\sigma$-coordinates.** One system of coordinates that is convenient for use in geophysics is known as the $\sigma$-coordinate system.\(^{26}\) The vertical coordinate variable $\sigma(x, z, t)$, which replaces $z$, is defined as

$$\sigma = \frac{z - \zeta}{H} = \frac{Z}{H} \tag{15}$$

\(^{26}\)Another is the isopycnal coordinate system (Higdon and de Szoeke, 1997).
(Blumberg and Mellor, 1987). Thus, $\sigma$ ranges from zero at the free surface to minus one at the bed. In contrast to $(x, z)$, the coordinate system $(x, \sigma)$ is in general non-orthogonal and non-stationary.

**Primitive equations.** The *primitive* equations of oceanic motion (Müller, 2006, §13) comprise a particular set of continuum balance equations, constitutive relations, and simplifying assumptions. These include the shallow-water approximation (essentially the boundary-layer approximation), and the Boussinesq approximation, comprising: (i) the anelastic approximation, which eliminates acoustic waves by removing the time derivative from the pressure equation; and (ii) the use of a reference density wherever mass density appears as a factor (Müller, 2006, §11). The pressure $p$ then becomes identified with the hydrostatic pressure $p_h$ (14).

6 Energy Extraction Modelling

There are (at least) three distinct ways in which the drag force due to an MRE device may be represented in the linear momentum balance equation (6). (i) *Momentum sink*: the drag force is modelled by adding a term to the specific body force vector $f$. (ii) *Actuator disk*: the device is modelled as a singular surface $S$, and the drag force is represented by the surface density $\varsigma_S$ of production of momentum. (iii) *Bed stress*: the device is modelled as an area of increased bed roughness, so that the drag force appears as an enhanced shear stress at the bottom boundary. These options are briefly described below.

6.1 Momentum Sink

In the ‘momentum sink’ approach, the drag is modelled as a distributed body force, which contributes to the momentum balance through the source term, represented in equation (6) by the body force density vector $\rho f$.  

23
Suppose the total drag force due to a tidal energy device is \( F_T(t) \). If it is assumed that this force acts uniformly over a spatial region \( V_T(t) \), then the corresponding specific drag force field is:

\[
f_T(x, t) = \frac{\nu_T(x, t)F_T(t)}{\rho(x, t)V_T(t)},
\]

where \( \nu_T \) is the indicator function of \( V_T \).\(^{27}\) More generally, \( \nu_T \) or \( \nu_T/(\rho V_T) \) may be defined as some other (ideally smooth) distribution function. The vector field \( f_T \) is added to the other terms that contribute to the total specific body force vector \( f \) on the right-hand side of equation (8). If multiple devices are to be represented as a single, aggregated value (as when the mesh is coarse), then the model should take into account any likely interference effects.

Yang et al. (2013) have implemented this method in FVCOM. The drag force is decomposed into three parts, to allow for the independent representation of the supporting structure, the foundations, and the rotor. Each device is considered to lie within a single control volume. The performance of the module has been tested for a tidal channel and bay system, and validated against the one-dimensional analytical model of Garrett and Cummins (2005) and Blanchfield et al. (2008) (see appendix A).

### 6.2 Actuator Disk

The concept of the actuator disk was originally developed for the analysis of flows through propellers and windmills.\(^{28}\) The approach has since been generalized such that it may be applied to the analysis of any kind of discontinuity in an otherwise continuous motion.

\(^{27}\)\( \nu_T(x, t) \) is equal to one if \( x \) belongs to \( V_T(t) \), and zero otherwise.

\(^{28}\)“The use of the actuator disk in engineering fluid mechanics dates back to the Rankine-Froude theory of the flow through a ship propellor. The actuator disk, which we define as an artificial device producing sudden discontinuities in flow properties, has since been used in numerous other engineering applications; to describe flow through turbine and compressor blade rows, through gauzes or screens, and through flame fronts.” (Horlock, 1978, §1)
With this method, the drag force is represented by the difference in the stresses acting on either side of a singular surface (see §5.1). The method works by imposing a jump condition (5) on the linear momentum balance. Writing $\psi = \rho v$ and $\varphi = -\sigma$ as in §5.2, and assuming the location of the device is fixed ($v_S = 0$), the jump condition is

$$n_S \cdot \left[ \rho \mathbf{v} \otimes \mathbf{v} - \sigma \right] = f_S,$$

where $f_S$ is the nett stress exerted on the fluid by the surface $S$. In one-dimensional flow this means that the thrust is balanced by a jump in the impulse stress $p + \rho v^2$ where $p$ is the pressure and $v$ the flow speed (Horlock, 1978, §2).

The actuator disk method has been implemented by Draper et al. (2010) using the discontinuous Galerkin finite element solver AD-CIRC, and by Roc et al. (2013) using the finite volume solver ROMS (Shchepetkin and McWilliams, 2005).

6.3 Bed Stress

The depth integral of the linear momentum balance equation is given by equation (9), with $\psi = \rho v$, $\varphi = -\sigma$, and $\varsigma = \rho f$ (see §5.2). The bottom boundary flux $\varsigma_B$ (10) has an advective part $V_B(\rho v)_b$, associated with the volume flux across the boundary, and a non-advective part $f_B = - (\nabla B) \cdot \sigma_b$, representing the bottom stress acting on the fluid, as a force per unit plan area. In practice, this stress is imposed as a bottom boundary condition in the linear momentum balance. In FVCOM, the bottom drag coefficient is computed as

$$C_d = \max (C_{d,\min}, C'_d), \quad \text{where} \quad C'_d = \left[ \frac{\kappa}{\ln(z_c/z_0)} \right]^2$$

(16)

is the nominal drag coefficient for a rough wall: $\kappa$ is von Kármán’s constant, $z_c$ is some predetermined height above the bed (namely, that of the centroid of the control volume in the bottommost $\sigma$-layer), and $z_0$ is the bottom roughness parameter (Chen et al., 2013,
The lower bound $C_{d,\text{min}}$ is a user-supplied constant value. The horizontal shear stress vector is then computed as

$$\tau_B = -C_d \rho_c |v_c| v_c,$$

where $\rho_c$ is the fluid mass density,¹²⁹ and $v_c$ is the horizontal velocity vector field at the vertical location $z_c$.⁴₀

If an MRE device is mounted on the bed, and the space it occupies lies entirely within the bottommost $\sigma$-layer, then it may be regarded as a localized area of enhanced roughness. The roughness height and the size of the affected area can be calculated such that the nett additional drag force corresponds to the estimated drag attributable to the device (assuming this is proportional to the square of the velocity). In principle, a similar approach could be taken for floating devices, with the drag force represented as an additional wind-shear stress at the top surface.

In a depth-averaged model, the bottom flux $\varsigma_B$ is equivalent to an additional internal source term, as discussed in §5.3. The momentum-sink (§6.1) and bed-stress methods in this case are equivalent.

The practical advantage of the bed-stress method is that it may be applied using existing models without modification. Also, if the solver has a turbulence model with production and dissipation terms at the wall, then, in principle, some of the effects of a device on the turbulence of the flow might be represented by means of the existing turbulence model.

¹²⁹$C_d$ is defined as $|\tau_B|/(\rho_c |v_c|^2)$: there is no factor $\frac{1}{2}$ in the denominator—cf. equation (1). The mass density $\rho_c$ in equation (17) is absent from the formula given by Chen et al. (2013).

⁴₀This is consistent with boundary layer theory. Outside the ‘viscous sub-layer’, dimensional analysis suggests that the mean velocity gradient is inversely proportional to the distance from the wall (Bradshaw, 1971, §3). This is essentially Prandtl’s mixing-length hypothesis (see Schlichting, 1961). It follows that the wall shear stress in bounded shear flow is given by equations (16)₂ and (17).
6.4 Secondary Effects

As discussed in §4, the secondary effects of energy extraction include the production and dissipation of turbulence and vorticity. The modelling of turbulence follows a similar pattern to the above: the turbulence energy and dissipation or length-scale balance equations take the place of the linear momentum equation, and the effects of MRE devices are modelled as distributed source (§6.1) or surface production (§6.2) terms (e.g. Roc et al., 2013).

7 Finite Volume Method

In the finite volume method, the flow domain is sub-divided into a finite number of non-overlapping cells, which collectively form a grid, or mesh. Finite volumes, called control volumes (CVs), are formed from these cells in various ways; and the governing equations are formulated as integral balance equations, with the CVs as domains of integration. Discrete state variables are usually defined as volume averages over the CVs. For the purposes of interpolation, these values may be associated with specific points called nodes. In general, each balance equation may have its own set of CVs and nodes.

7.1 Spatial Discretization

In the horizontal domain, FVCOM uses an irregular, unstructured grid of triangular cells (Chen et al., 2003). There are two distinct sets of nodes and associated CVs.

(i) For the purposes of horizontal momentum balance, the CVs are the cells, and the nodes are located at cell centroids. Horizontal velocity components are stored at these nodes.

(ii) All other variables are stored at cell vertices. Scalar CVs are constructed by joining the mid-point of each edge connected to a given vertex, to the centres of the two cells adjacent to
that edge. The result is a star-shaped polygon whose vertices are alternately the centroids of the connected cells and the mid-points of the connected edges. A cell vertex may be connected to any number (typically between five and eight) triangular cells (see figure 3).

For the vertical discretization, FVCOM has a number of options. The original version uses the $\sigma$-coordinate system (see §5.3). The vertical domain is divided into a number of $\sigma$-layers. The surfaces which separate them are called $\sigma$-levels. Vertical components of velocity, and state variables associated with turbulence, are stored at $\sigma$-levels; all other variables are stored at mid-points of $\sigma$-layers (Chen et al., 2013, §3). The vertical location and extent of a CV vary, but its $(\mathbf{x}, \sigma)$ coordinates are fixed.

### 7.2 Discrete Balance

The semi-discrete (i.e. spatially discretized) form of the general balance equation (4) can be obtained by transforming from $(\mathbf{x}, z)$ to $(\mathbf{x}, \sigma)$ coordinates (15) and then integrating with respect to the fixed limits that define the spatial extent of a given CV. The transformation is necessary because the $(\mathbf{x}, z)$ coordinates of a CV vary with time. Alternatively, we may apply the depth-integration procedure of §5.3 to each individual $\sigma$-layer, and then integrate in the usual way with respect to the fixed horizontal coordinates of a CV.

Suppose there are $n$ $\sigma$-layers, with $\sigma$-levels numbered from $i = 0$ at the top surface to $i = n$ at the bottom. Let $\sigma_i$ denote the $\sigma$-coordinate of the $i^{th}$ level. From equation (15), $\sigma_0 = 0$ and $\sigma_n = -1$. The $i^{th}$ layer extends from $\sigma = \sigma_i$ at its lower surface to $\sigma = \sigma_{i-1}$ at its upper, and its depth is given by

$$ H_i(\mathbf{x}, t) = (\sigma_{i-1} - \sigma_i)H(\mathbf{x}, t) = z_{i-1}(\mathbf{x}, t) - z_i(\mathbf{x}, t), $$

---

31 If all the cells surrounding a cell vertex were equilateral triangles, the scalar CV would form a regular hexagon with vertices at the cell centroids.
where \( z_i = \zeta + H\sigma_i \) is the \( i \)th \( \sigma \)-level. We assume that the inter-layer fluxes \( \varsigma_i \) \((0 < i < n)\) are conservative, i.e. there is no surface production on the inter-layer boundaries. Then \( \varsigma_i \) represents simultaneously the influx through the lower boundary of layer \( i \) and the outflux through the upper boundary of layer \( i + 1 \).

**Layer-average.** Let \( \overline{\psi}_{(i)} \) denote the depth-averaged value of \( \psi \) over the \( i \)th layer, i.e.

\[
\overline{\psi}_{(i)}(x, t) = \frac{1}{H_i(x, t)} \int_{z_i(x, t)}^{z_{i-1}(x, t)} \psi(x, z, t) \, dz.
\]

The layer-integrated balance equation for \( \psi \) in this layer is obtained directly from equation (9):

\[
\frac{\partial H_i \overline{\psi}_{(i)}}{\partial t} + \nabla \cdot \left( H_i (v \otimes \psi)_{(i)} + H_i \varphi_{(i)} \right) = H_i \varsigma_{(i)} - \varsigma_{i-1} + \varsigma_i, \tag{18}
\]

where: \( \varsigma_i = V_i \psi_i + (\nabla Z_i) \cdot \varphi_i \) is the flux density (10) from layer \( i + 1 \) (or the bed if \( i = n \)) to layer \( i \) (or through the top surface if \( i = 0 \)); \( V_i \) is the corresponding volume flux density; \( Z_i(x, z, t) = z - z_i(x, t) \) is the scalar field whose zero-contour is the \( i \)th \( \sigma \)-level; and

\[
\psi_i(x, t) = \psi(x, z_i(x, t), t) \quad \text{and} \quad \varphi_i(x, t) = \varphi(x, z_i(x, t), t)
\]

are the field-variable values at that level. If \( n = 1 \), equations (9) and (18) are identical.

**Control-volume average.** The spatially discrete balance equation for a given CV is obtained from equation (18) by integrating over the fixed horizontal region, \( A \), occupied by the CV. The integral of the time-derivative term may be written

\[
\int_A \frac{\partial H_i \overline{\psi}_{(i)}}{\partial t} \, dA = \frac{\partial V_i [\psi]_{(i)}}{\partial t},
\]
where $V_i$ is the volume of the CV (in the $i$th layer), and $[\psi]_i$ is the average value of $\psi$ in that CV:

$$V_i(t) = \int_A H_i(x, t) \, dA, \quad [\psi]_i(t) = \frac{1}{V_i(t)} \int_A H_i(x, t) \bar{\psi}_i(x, t) \, dA.$$ 

Similarly, the integral of the source term $H_i \zeta_i$ is $V_i \zeta_i$.

The integral of the divergence term in equation (18) may be expressed as a boundary-integral using Green’s theorem (cf. §5.1):

$$\int_A \nabla \cdot \left( H_i (v \otimes \psi)_i + H_i \varphi_i \right) \, dA = \int_{\partial A} H_i \left( \bar{n}^n_i \bar{\psi}_i + n \cdot \bar{\varphi}^* \right) \, dL,$$

where $\partial A$ denotes the lateral boundary of $A$, $n(x)$ is the outward-normal dual vector field, defined on $\partial A$, $\varphi^*$ is the effective flux tensor (11), $L$ is the length measure, and

$$\bar{n}^n_i = n \cdot v_i$$

is the layer-average normal velocity component, defined on $\partial A$. In FVCOM, $A$ is a polygon, and the integral over $\partial A$ is evaluated as a finite sum, with one value for each edge. In constructing the discrete balance equation, the average value of a flux over an edge is approximated numerically using field variable values at nodes.

### 7.3 Numerical Solution

The terms above combine to give a discrete balance equation for each CV. This gives a system of coupled ordinary differential equations involving CV-average values of the state variables, lateral boundary fluxes, and top and bottom surface fluxes. FVCOM solves these equations numerically using a mode-split procedure. The depth-averaged equations of motion are solved first, using a relatively small time-step, in order to resolve the relatively fast motions associated with the external mode. This two-dimensional solution is used to advance the three-dimensional solution, using a larger time-step,
chosen to accommodate the slower motions of the internal mode. Both systems are solved using a second-order, four-stage, explicit Runge-Kutta method with a fixed time-step (Chen et al., 2003).

8 Implementation

The energy extraction module is designed as a user-programmable Fortran 95 module, to be compiled as part of FVCOM. Its main purpose is to provide a procedure that computes the drag force for a CV or CVs, given the instantaneous values of the local state variables as input arguments. Thus, a wide range of empirical drag formulae can be implemented without altering the module’s interface.

The most general implementation would take account of how the total drag force is spatially distributed. In principle, the drag function could be implemented without reference to the structure of the computational mesh, so that the same function could be used in other programs. Information on the spatial distribution of the device could be used to determine, for each CV, the fraction of the total force associated with that CV (as an average force density).

A simpler alternative, adequate for our purposes, is to suppose that each MRE device is contained within a single CV, so that the drag force is represented by a single term in the balance equation for that CV. This is the approach taken by Yang et al. (2013). The drag function, as an average force density, still depends on the mesh geometry, but only through a dependence on the volume—not the shape or location—of the CV containing the device.

Since FVCOM employs $\sigma$-coordinates in the vertical direction (see §7.1), the volume and vertical location of a cell vary with time, following the motion of the free surface. If the drag force due to an individual device were distributed over several CVs, it would be necessary to re-compute the spatial distribution continually. In the proposed, simpler implementation, no adjustment is necessary to the vertical location, as long as the vertical translational motion of the CV is small. Note, however, that, for a given total force, an
Figure 3: Idealized channel mesh and bathymetry. Colour represents bed elevation plus bed roughness height (m).

increase in the volume of a CV implies a corresponding decrease in the average force density.

9 Idealized Channel Simulation

An idealized channel flow problem was set up to serve as a test case for the preliminary evaluation of energy extraction modelling in FVCOM. The dimensions of the channel were chosen to be roughly representative of the Pentland Firth (see figure 1). The horizontal mesh is shown in figure 3. The channel itself occupies the central region of the model domain, between the $x$-coordinate values of zero and 20.4km. The width of the channel varies linearly from 12.7km at the western end (representing a line from the southern tip of Hoy to Dunnet Head) to 10.2km at the eastern end (representing a line from the southern tip of South Ronaldsay to Duncansby Head). The mean depth varies linearly from 90m in the west to 70m in the east. The mesh has three $\sigma$-layers; the resulting vertical resolution is coarse,
but adequate for the purposes of illustration. The computational mesh extends beyond the nominal ends of the channel to include an expansion region with a horizontal bed, leading to a curved open boundary, at each end.\textsuperscript{32}

The open boundaries are forced with prescribed tidal elevations using the single constituent \( M_2 \) (period 12.42h). The amplitudes of the forcing are 1.18m at the western, and 0.81m at the eastern boundary, which has a prescribed phase lag of 76.2° relative to the western boundary. Sponge layers are specified at all open boundary nodes to reduce computational noise, as suggested in the FVCOM manual (Chen et al., 2013, §6).\textsuperscript{33} The bottom roughness parameter is set to a small value (1mm), so that the drag coefficient \( C_d \) at the bed takes its default minimum value \( C_{d, \text{min}} = 0.0025 \).

We consider two MRE sites, each approximately 5km from the nominal ends of the channel, lying on opposite sides of the centre-line, as shown in figure 3. In the demonstration simulation presented here, these sites are modelled as regions of enhanced bed roughness (see §6.3). A roughness height of 3m was chosen for this study, which may be high, but serves to illustrate clearly the effects.

Tests were run using FVCOM both in normal three-dimensional mode and in two-dimensional, depth-averaged mode.\textsuperscript{35} The baseline case (without energy extraction) has a uniform, default bed roughness height of 1mm. The computed ‘west-east’ velocity fields from the three-dimensional simulation for this case are shown in figure 4.

\textsuperscript{32}The extended boundary regions were added for numerical stability purposes.

\textsuperscript{33}The topic of open boundary treatments is an active area of research. A recent review of the various kinds of boundary conditions applied in ocean modelling is given by Herzfeld et al. (2011). These include ‘viscosity sponges’, where “[t]he horizontal viscosity (and diffusion) coefficient may be increased over a sponge zone adjacent to the boundary so that outgoing transient waves are damped before reaching the boundary.”

\textsuperscript{34}The bottom drag coefficient is given by equation (16). If \( \kappa = 0.4 \) and \( z_0 = .001 \) then \( C_d = C_{d, \text{min}} \) whenever \( z_c \) is greater than 3m. In our case \( z_c \) is everywhere greater than 10m.

\textsuperscript{35}FVCOM is written with a number of C-preprocessor flags that cause different blocks of source code to be either included or excluded at compile-time. The two-dimensional model is obtained by setting the TWO_D_MODEL flag.
Figure 4: Idealized channel: base-line velocities (m/s) just before peak flood tide.
Figure 5: Idealized channel with energy extraction: velocities (m/s) just before peak flood tide.
at a time instant just prior to peak flood tide (eastward flow). The corresponding plots, at the same instant, in the energy extraction case are shown in figure 5. Similar plots for the ebb tide (half a cycle later) are shown in figures 6 and 7. The narrowing of the channel towards the eastern end, combined with the fact that the channel is shallower there, means that the flow speed is generally highest at the eastern end, and the pattern of flow on the ebb tide is quite different to that on the flood.

The bottom-layer and depth-averaged velocities are greatly reduced at the MRE sites, as expected. There is also significant alteration to the surrounding flow, and this is not confined to the wake. Due to their asymmetrical placing and relatively high resistance, some flow is diverted away from the MRE sites, giving increased velocities in neighbouring areas. It is clear that the cross-sectional average flow speeds are not representative of local site conditions. The overall effect of these asymmetrically-placed sites is qualitatively quite different to the ‘turbine fence’ scenario considered in a number of other studies (e.g. Adcock et al., 2013).

These preliminary results show that a reliable assessment of the performance of a real MRE installation in situ requires a model that is capable of resolving the altered flow conditions without assuming either vertical or cross-channel uniformity. Note that, while a given three-dimensional model is more accurate than its two- or one-dimensional counterpart, any discretized model can be improved by better accounting for the effects of the sub-integral-scale motion. Thus, a ‘two-dimensional’ finite volume model in which the vertical variations are accurately represented may actually give better results than a standard three-dimensional model.

9.1 One-dimensional Model

One-dimensional models have proved useful in estimating the effects of energy extraction in tidal channel networks, for example (Polagye et al., 2009). Rigorous inter-comparison of one- and three-dimensional models requires consideration of the ‘deviation fluxes’
Figure 6: Idealized channel: base-line velocities (m/s) just before peak ebb tide.
Figure 7: Idealized channel with energy extraction: velocities (m/s) just before peak ebb tide.
which arise due to averaging (see §5.3). An account of an established one-dimensional approach to modelling is given in appendix A.

A channel modelling program was developed for the purpose of comparison with FVCOM, based on an existing program that solves the linearized Saint Venant equations.\textsuperscript{36} The discretized equations are solved using leap-frog time-stepping with an Asselin filter (see Shchepetkin and McWilliams, 2005).

The linearized model predicted symmetrical flows for the idealized channel problem, i.e. with velocities varying around a mean value of zero. In contrast, the FVCOM results display a pronounced asymmetry, with flow on the ebb tide exceeding that on the flood. With the non-linear terms included, the one-dimensional model also predicts asymmetrical flows.

There are, however, substantial differences between the results. To explain these differences, the investigation focused initially on the open boundary treatment. It became clear that the major cause of the discrepancy was due to inaccurate specification of tidal boundary data. In particular, the one-dimensional results are very sensitive to changes in the phase difference between the two open boundaries (see Carter and Merrifield, 2007). In the event, a rigorous comparison was not possible in the time available.

10 Conclusions

\begin{enumerate}
\item Detailed assessment of the performance and potential hydrodynamic effects of MRE installations requires the use of computational ocean dynamics models. Methods for the representation of the effects of MRE devices are at an early stage of development, and their validity is largely unknown.

\item Ocean models are based on the balance equations of continuum physics. Precise forms of these equations have been derived,
\end{enumerate}

\textsuperscript{36}The linearized momentum equation is obtained by neglecting the advective transport term $u\partial u/\partial x$ in equation (19).
in general tensor form, for arbitrary depth-averaged flows and cross-section-averaged channel flows. The layer-averaged equations, for models using $\sigma$-coordinates, are obtained by the same method. The form of these equations is ideally suited to the formulation of finite-volume solution methods. They are valid for the transport of any extensive tensor property, including volume, momentum, mass, energy, turbulence, salinity, etc.

(iii) MRE devices can be represented by any of the momentum sink, actuator disk, and bed stress methods. In depth-averaged models, the momentum sink and bed stress methods are equivalent. The latter is appropriate for bottom-mounted devices in three-dimensional models. In principle, the same method could be applied at the free surface, to represent floating devices by means of enhanced wind stress.

(iv) Simulations of an idealized channel have shown the effect of two hypothetical MRE sites, placed asymmetrically on the bed. A complex asymmetrical flow pattern results, and it appears that accurate estimates of MRE performance in such cases requires the use of a model that can resolve the vertical and cross-channel variability.

References


Appendix

A One-Dimensional Modelling

Here we present an approach to the representation of MRE devices in the context of one-dimensional flow modelling. Emphasis is placed on identifying some of the key assumptions.

Garrett and Cummins (2005) consider a convergent–divergent channel connecting two basins. The analysis begins with the momentum balance equation for simple one-dimensional flow:

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + g \frac{\partial \zeta}{\partial x} = -F, \tag{19}
\]

where \(u(x,t)\) is the flow velocity, \(\zeta(x,t)\) is the free-surface elevation, \(F(x,t)\) is a specific body force (i.e. force per unit mass) representing the net effect of the bed stress and the drag due to any MRE devices,
$x$ denotes location along the channel axis, $t$ is time, and $g$ is the gravitational constant. Implicit in this formulation is an assumption that the drag can be represented as a distributed body force (see §6). Assuming no lateral inflows, the continuity equation may be written

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0,$$

(20)

where $A(x, t)$ is flow cross-sectional area, and $Q = Au$ is discharge.

Garrett and Cummins (2005) restrict attention to cases where “the channel is short compared with the wavelength of the tide”. Then the discharge $Q$ is (approximately) independent of $x$. In accordance with equation (20), it is also assumed that the flow area $A$ is independent of $t$. Then we may write

$$u(x, t) = \frac{Q(t)}{A(x)}.$$

At the channel entrance, it is supposed that the flow is “drawn in smoothly from a region with large $A$, weak currents and prescribed tidal elevation.” The velocity head at the entrance may then be neglected. It is assumed that the flow emerges from the (divergent) channel into the basin in the form of a jet. The location of the channel ‘exit’—which determines the nominal length $L(t)$ of the channel—is then defined as the point where the flow separates from the channel walls. Integrating the momentum equation (19) over the length of the channel, from $x = 0$ to $x = L$, gives

$$c \frac{dQ}{dt} = g\zeta_0 - \frac{1}{2}|u_e|u_e - \int_0^L F \, dx,$$

(21)

where $c(t) = \int_0^{L(t)} A^{-1} \, dx$ is a geometric parameter, $u_e(t)$ is the exit velocity, and $\zeta_0(t) = \zeta(0, t) - \zeta(L, t)$ is the difference in sea level between the two basins. The parameter $c$ is assumed constant.

37 The ‘entrance’ switches periodically from one end of the channel to the other as the flow changes direction.

38 The exit velocity $u_e$ is equal to $u(0, t)$ if $Q(t) < 0$, and $u(L, t)$ otherwise.
The specific drag force $F$ is defined as a function of the coordinates $(x, t)$; hence it may be a function of any of the state variables $u(x, t)$, $A(x)$, etc. Given a specific functional form for $F$, equation (21) can be solved (numerically or analytically), for a given channel geometry ($A$ and $L$) and prescribed tidal forcing ($\zeta_0$), to obtain the discharge $Q(t)$ at any point $t$ in the tidal cycle.

The total power lost by the flow due to the action of $F$ over the region $x = 0$ to $x = L$ is given by

$$P(t) = \rho Q(t) \int_0^L F(x, t) \, dx,$$

where $\rho$ is the fluid mass density (assumed constant). This may be written as a sum $P = P_b + P_T$, corresponding to a decomposition of $F$ into terms, $F_b$ and $F_T$, attributable to the bed and to MRE devices respectively. The power $P_T$ lost by the flow due to the presence of MRE devices is then given by equation (22) with $F_T$ in place of $F$. Not all of this power is useful: there are kinetic energy losses, both locally and in the wake.

The average power lost by the flow over a full tidal cycle is

$$\overline{P} = \frac{1}{T} \int_0^T P \, dt,$$

where $T$ is the tidal period. Considering a general form of drag law in which the drag force is proportional to $u^n$, Garrett and Cummins (2005) computed $\overline{P}$ for a range of values of $n$ between $\frac{1}{4}$ and 3. Their results show that $\overline{P}$ exhibits a maximum value, $\overline{P}_{\text{max}}$, representing a point beyond which any increase in $F$ (corresponding to an increase in turbine density) causes a reduction in total power, due to choking. Regarding the exponent $n$ as a variable, it is found that the greatest value of $\overline{P}_{\text{max}}$ is obtained when $n = 1$; therefore, this value can be taken as a convenient upper bound on the available energy resource.