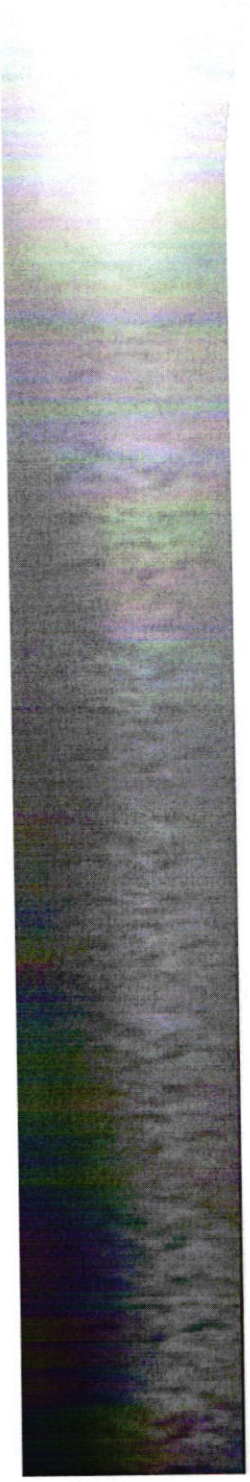

BEST PRACTICE GUIDELINES

FOR

MARINE APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS

MARNET CFD

Prepared by
WS Atkins Consultants
And members of the NSC

Sirehna
HSVA
FLOWTECH
VTT
Imperial College of Science & Technology
Germanischer Lloyd
Astilleros Espanoles

MARCHI
SET/03

HTTPS://PRONET.WSATKINS.CO.UK/MARNET/PUBLICATIONS/BPG.PDF

Table of Contents

1. INTRODUCTION **6**

1.1. Background **6**

1.2. Scope **6**

1.3. Structure of this document **7**

1.4. Acknowledgements and other sources **7**

2. OVERVIEW OF EQUATIONS AND METHODS IN MARINE CFD **8**

2.1. Fluid equations of motion **8**
2.1.1. General Fluid Dynamic Equations 8
2.1.2. The Assumption of Incompressibility 9
2.1.3. Turbulence 10
2.1.4. Potential Flow 11

2.2. General boundary conditions **12**

2.3. Coupling with motions of floating systems **12**
2.3.1. General comments 12
2.3.2. Linear Harmonic Non-steady Problems 13
2.3.3. Non-Linear and Time Domain Simulations 13

2.4. Boundary element or panel methods for potential flow **14**

2.5. Methods for viscous turbulent flows **16**
2.5.1. General 16
2.5.2. Finite difference method 16
2.5.3. Finite element method 17
2.5.4. Spectral method 17
2.5.5. Finite volume method 17

2.6. Dealing with the free surface **18**
2.6.1. Potential flow 18
2.6.2. Viscous flow 19

3. GENERAL ERRORS AND UNCERTAINTIES IN CFD SIMULATIONS **21**

3.1. Sources of errors and uncertainties and their classification **21**
3.1.1. Model error and uncertainty 21
3.1.2. Discretisation or numerical error 22
3.1.3. Iteration or convergence error 22
3.1.4. Round-off errors 22
3.1.5. Application uncertainties 22
3.1.6. User Errors 22
3.1.7. Code Errors 22

3.2. Definitions of errors and uncertainties **23**

3.3. Definitions of verification, validation and calibration **23**

4. METHOD INDEPENDENT ERRORS AND UNCERTAINTIES - GUIDELINES **24**

4.1. Convergence errors **24**

4.2. Round-off errors **25**

4.3. Spatial discretisation errors **26**

4.4. Temporal discretisation for unsteady problems **27**

4.5. Geometrical uncertainties **28**

4.6. User errors **28**
4.6.1. General comments 28
4.6.2. Control of the working process 29
4.6.3. Training requirements for CFD users 30

4.7. Code errors **31**

5. POTENTIAL FLOW AND DIFFRACTION CALCULATIONS **33**

5.1. General guidance in panel mesh generation **33**

5.2. Definition of boundary conditions **34**

5.3. Special considerations for non-linear methods **35**

5.4. Integration of viscous effects **35**
5.4.1. General 35
5.4.2. Empirical and semi-empirical methods 35
5.4.3. Solution of Navier Stokes equations 35

6. VISCOUS TURBULENT FLOWS **37**

6.1. Solution algorithm **37**

6.2. Turbulence modelling **37**
6.2.1. RANS equations and turbulence models 38
6.2.2. Classes of turbulence models 38

6.3. Weaknesses of the standard k-ε model **41**
6.3.1. Guidelines on weaknesses of the standard k-ε model 41

6.4. Near wall modelling **41**
6.4.1. Wall functions 42
6.4.2. Wall function guidelines 42
6.4.3. Near wall resolution 42
6.4.4. Near wall resolution guidelines 43

6.5. Inflow boundary conditions **43**

6.6. Unsteady flows **43**

6.7. Laminar and transition flows **43**
6.7.1. Guidelines 43

6.8. Mesh generation **43**

6.9. Choice of boundary conditions **45**

6.10. Application of boundary conditions **45**
6.10.1. General guidelines on boundary conditions 46
6.10.2. Guidelines on inlet conditions 46
6.10.3. Guidelines on specification of turbulence quantities at an inlet 46
6.10.4. Guidelines on outlet conditions 47
6.10.5. Guidelines on solid walls 47
6.10.6. Guidelines on symmetry and periodicity planes 47

6.11. Steady flow, symmetry, periodicity, etc. **47**
6.11.1. Guidelines 48

6.12. Analysis of results, sensitivity studies and dealing with uncertainties 48
6.12.1. Analysis of results 48
6.12.2. Sensitivity studies 49
6.12.3. Dealing with uncertainties 49

7. APPLICATION EXAMPLES 50

7.1. Example of Wave Pattern Calculations for Steady Ship Flow 50
7.1.1. Introduction 50
7.1.2. Geometry and boundary conditions 50
7.1.3. Grid 51
7.1.4. Features of the simulation 52
7.1.5. Results 52
7.1.6. Conclusions 54
7.1.7. References 54

7.2. Example of viscous stern flow calculations 55
7.2.1. Introduction 55
7.2.2. Geometry and boundary conditions 55
7.2.3. Features of the simulation 58
7.2.4. Results 58
7.2.5. References 59

7.3. Example of Unsteady Manoeuvring Calculations 59
7.3.1. Introduction 59
7.3.2. Geometry and boundary conditions 60
7.3.3. Grid 60
7.3.4. Results 61
7.3.5. Identification of errors and uncertainties 63
7.3.6. Conclusions 63
7.3.7. References 64

7.4. Example of propeller flow calculations 65
7.4.1. Introduction 65
7.4.2. Geometry 65
7.4.3. Grid 65
7.4.4. Features of the Simulation 68
7.4.5. Results 69
7.4.6. References 73

8. CHECKLIST OF BEST PRACTICE ADVICE FOR MARINE CFD 74

8.1. General CFD guidelines 74
8.1.1. Guidelines on the training of CFD users 74
8.1.2. Guidelines on problem definition 74
8.1.3. Guidelines on global solution algorithm 74
8.1.4. Guidelines on the solution of the discretised equations 75
8.1.5. Guidelines on assessment of errors 75
8.1.6. Guidelines on analysis and interpretation of results 76
8.1.7. Guidelines on documentation 76
8.1.8. Guidelines on communication with code developer 77

8.2. RANS calculations 77
8.2.1. Guidelines on solution strategy 77
8.2.2. Guidelines on turbulence modelling 78
8.2.3. Guidelines on definition of geometry 79
8.2.4. Guidelines on grids and grid design 79
8.2.5. Guidelines on boundary conditions 80
8.2.6. Guidelines on convergence 82

8.3. Potential flow **83**
8.3.1. Guidelines on definition of non-linear problems 83
8.3.2. Guidelines on integration of viscous effects 83
8.3.3. Guidelines on definition of geometry 83
8.3.4. Guidelines on boundary conditions 84

9. REFERENCES **853**

1. Introduction

1.1. Background

The availability of robust commercial computational fluid dynamics (CFD) software and high speed computing has lead to the increasing use of CFD for the solution of fluid engineering problems across all industrial sectors and the marine industry is no exception. Computational methods are now routinely used, for example, to examine vessel boundary layer and wake, to predict propeller performance and to evaluate structural loads.

Recently there has been a growing awareness that computational methods can prove difficult to apply reliably i.e. with a known level of accuracy. This is in part due to CFD being a knowledge-based activity and, despite the availability of the computational software, the knowledge base embodied in the expert user is not available. This has lead to a number of initiatives that have sought to structure existing knowledge in the form of best practice advice. Two notable examples are the best practice guidelines developed by ERCOFTAC and the European Thematic network QNET-CFD. The guidelines presented here build on the work of these two initiatives, particularly the ERCOFTAC BPGs, which with some modification and adaptation, have been used as a template for these guidelines.

The guidelines provide simple practical advice on the application of computational methods in hydrodynamics within the marine industry. It covers both potential and viscous flow calculations.

The range of CFD tools available for these classes of problem is broad and varied. Furthermore, their development has followed different paths, with both specialised maritime CFD packages and more general engineering CFD tools being applied to these problems. This has presented somewhat of a problem in developing these guidelines. However, it is true to say that there are many common elements regardless of the tools being used. The need to understand the physics of the problem in hand, the limitations of the equations being used, the basis of the numerical methods employed and the means to get the most accurate and consistent results for the available computing resource, are but some of the common challenges faced by the CFD user in maritime sector with his or her counterparts in other fields of engineering.

These guidelines therefore address these common aspects of CFD. Problem specific guidance, relating to phenomena such as cavitation on propellers or green water wave loading on offshore structures, are covered in the accompanying Application Guidelines which are being developed within each of the MARNET-CFD Thematic Area Groups.

1.2. Scope

This document provides both background and guidance for the methods used to examine flows which are incompressible, steady and unsteady, laminar and turbulent with or without free surfaces. The guidelines address both potential and viscous flow methods, and the aspects of CFD that are common to all methods.

These advice presented is relevant to problems involving:

- vessel boundary layers and wakes
- seakeeping
- vessel manoeuvring
- propeller performance
- control surface performance
- fluid/structure interaction
- offshore fluid loading and floating platform response
- free surface flow

1.3. Structure of this document

Following this introduction, an overview of the general methods used in marine CFD is presented. This begins with a review of the fluid equations of motion and the ways in which they are used, and then examines the theories behind potential and viscous flow methods. Free surface flows and the specific ways of modelling them are also discussed.

This is followed by the definition of the concepts of general errors and uncertainties in CFD, and a comprehensive section providing guidelines on how to deal with method independent errors and uncertainties. Guidelines are given to draw the user's attention to the likely sources of uncertainty when formulating a problem, and the known sources of error inherent in CFD methods.

Detailed issues to be considered in modelling potential and viscous flows are then discussed, presenting the user with guidance aimed at making problem formulation and simulation easier and more accurate.

This is followed by a comprehensive section dealing with best practice guidelines for viscous incompressible turbulent flow calculations using RANS methods.

The section on application examples provides illustrations of some typical uses of CFD for the maritime environment, and illustrates many of the main points of the guidelines.

This is followed by a checklist of best practice guidance, designed to act as a quick reference section, and compiled as a summary of best practice advice given in the previous sections.

Finally, a section is included which provides a reference to typical general purpose and dedicated marine CFD codes for use in design assessment work in the marine industry.

1.4. Acknowledgements and other sources

These best practice guidelines have been compiled through input from each of the MARNET thematic area co-ordinators. We have also made use of the ERCOFTAC IAC best practice guidelines for viscous flow, from which areas relevant to marine hydrodynamics have been extracted (Chapters 3, 4 and 6). This has been done with a view to making a contribution to ERCOFTAC SIG 25 (ship hydrodynamics) and the QNET-CFD Thematic Network, which is itself establishing broad best practice guidelines in CFD for the whole of European industry. The latter will be working toward industry specific guidelines, and MARNET-CFD will add to this knowledge base.

Other resources include the works of authors from WEGEMT school lecture notes on maritime CFD and the Reports and Recommendations to the 22nd ITTC.

2. Overview of equations and methods in marine CFD

2.1. Fluid equations of motion

In marine CFD we are chiefly concerned with problems in hydrodynamics. In the majority of problems being solved, we are attempting to calculate global pressures and fluid velocity components in a 3 dimensional space surrounding the submerged portion of the marine vehicle or platform of interest. In this way, it is possible to further calculate the forces and moments acting on the vessel, whether steady or unsteady. It is customary to treat the working fluid, in this case water, as incompressible and isothermal. However, it is also possible to make further assumptions regarding the behaviour of the flow, depending upon the nature of the problem in hand and the leading order effects of interest.

Therefore here, we start from the beginning and provide definitions of the general fluid equations of motion, from which such special cases (such as gravity driven, incompressible, inviscid and irrotational free surface waves – potential flow) can be derived. The majority of commercial CFD software tools have been written to solve the more general cases of compressible, viscous, turbulent flows with heat transfer, but may be applied to problems in hydrodynamics, so long as the correct choices are made regarding equations of state, fluid properties, and boundary conditions. The definitions given below should provide those attempting problems in hydrodynamics with a guide to how the equations of most interest are derived.

2.1.1. General Fluid Dynamic Equations

The general equations of fluid flow represent mathematical statements of the conservation laws of physics, such that:

- Fluid mass is conserved
- The rate of change of momentum equals the sum of the forces on a fluid particle
- The rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on a particle.

The governing equations for an unsteady, three dimensional, compressible viscous flow are:

Continuity equation:

$$\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho U)=0 \qquad (1)$$

Momentum equations:

x component:

$$\frac{\partial(\rho u)}{\partial t}+\nabla\cdot(\rho u U)=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+\rho f_x \qquad (2)$$

y component:

$$\frac{\partial(\rho v)}{\partial t}+\nabla\cdot(\rho v U)=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho f_y \qquad (3)$$

z component:

$$\frac{\partial(\rho w)}{\partial t}+\nabla\cdot(\rho w U)=-\frac{\partial p}{\partial w}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z \qquad (4)$$

Energy equation:

$$\frac{\partial}{\partial t}\left(\rho(e+\frac{U^2}{2})\right)+\nabla\cdot\left(\rho U(e+\frac{U^2}{2})\right)=\rho\dot{q}+\frac{\partial}{\partial x}(k\frac{\partial T}{\partial x})+\frac{\partial}{\partial y}(k\frac{\partial T}{\partial y})+\frac{\partial}{\partial z}(k\frac{\partial T}{\partial z})$$
$$-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}$$
$$+\frac{\partial(u\tau_{xx})}{\partial x}+\frac{\partial(u\tau_{yxx})}{\partial y}+\frac{\partial(u\tau_{zxx})}{\partial z}$$
$$+\frac{\partial(u\tau_{xy})}{\partial x}+\frac{\partial(u\tau_{yy})}{\partial y}+\frac{\partial(u\tau_{zy})}{\partial z} \qquad (5)$$
$$+\frac{\partial(u\tau_{xz})}{\partial x}+\frac{\partial(u\tau_{yz})}{\partial y}+\frac{\partial(u\tau_{zz})}{\partial z}+\rho f U$$

where: ρ is the fluid density, $U = (u, v, w)$ the fluid velocity, p the pressure, T the temperature, e is the internal energy per unit mass, $f = (f_x, f_y, f_z)$ is a body force, k is the thermal conductivity, $\dot{q}$ is the rate of volumetric heat addition per unit mass and τ_{nn} are the viscous stresses.

These equations represent 5 transport equations in 7 unknowns, u, v, w, p, T, ρ and e. They are completed by adding two algebraic equations; one relating density to temperature and pressure:

$$\rho=\rho(T,p) \qquad (6)$$

and the other, relating static enthalpy to temperature and pressure:

$$h=h(T,p)\,. \qquad (7)$$

2.1.2. *The Assumption of Incompressibility*

For incompressible flow such as we require for hydrodynamics, and assuming that the fluid is Newtonian and that the viscosity is constant throughout the flow, the continuity equation becomes:

$$\nabla . U=0 \qquad (8)$$

The momentum equations become:

x component: $$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x}+\mu\nabla^2 u+\rho\cdot f_x \qquad (9)$$

y component: $$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y}+\mu\nabla^2 v+\rho\cdot f_y \qquad (10)$$

z component: $$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial z}+\mu\nabla^2 w+\rho\cdot f_z \qquad (11)$$

Where *D/Dt* is the substantial derivative given by:

$$\frac{D}{Dt}=\frac{\partial}{\partial t}+u\frac{\partial}{\partial x}+v\frac{\partial}{\partial y}+w\frac{\partial}{\partial z}. \qquad (12)$$

The continuity and momentum equations are now de-coupled from the energy equation and are all that is necessary to solve for the velocity and pressure fields in an incompressible flow.

2.1.3. Turbulence

Whilst the above equations are sufficient for the description of incompressible, laminar flow, and being a description of a continuum, in principle apply to all scales, they are also non-linear and subject to instability. Physically, these instabilities grow to provide a mechanism to describe turbulence. Practically, this renders the equations impossible to solve analytically, and requires that numerical methods be formulated to solve for particular (statistically stationary) states within the flow.

It is assumed that the components of the flow velocity, and the pressure, consist of a mean value with superimposed fluctuations. These fluctuations are bounded to remain within a spectrum of values in terms of frequency and amplitude. This spectrum of the turbulent kinetic energy can be analysed and operated on using statistical tools, from which a variety of formulations for the mass and momentum conservation can then be derived.

The most well known of these operations is known as Reynolds averaging, and forms the basis of the Reynolds-averaged Navier Stokes Equations (RANSE). The velocity components are represented by:

$$U=U(x)+U'(x,t) \qquad (13)$$

where $U(x)$ is the mean and $U'(x,t)$ is the unsteady disturbance quantities in the flow, such that $\overline{U'}=0$.

On time averaging, the x-component momentum equation becomes:

$$\rho\left[\frac{\partial(u^2)}{\partial x}+\frac{\partial(uv)}{\partial y}+\frac{\partial(uw)}{\partial z}\right]=-\frac{dP}{dx}+\frac{\partial}{\partial x}\left[\mu\frac{\partial u}{\partial x}-\rho\overline{u'^2}\right]+\frac{\partial}{\partial y}\left[\mu\frac{\partial u}{\partial y}-\rho\overline{u'v'}\right]+\frac{\partial}{\partial z}\left[\mu\frac{\partial u}{\partial z}-\rho\overline{u'w'}\right] \qquad (14)$$

The equations for the other components take a similar form. The Reynolds stresses ($\rho\overline{u'v'},\rho\overline{u'w'}$, etc.) are treated as extra stresses that arise from the turbulent nature of the flow.

The problem then arises to calculate these stresses. There are many ways in which this can be achieved, all relying to greater or lesser extents on further assumptions and simplifications. The resulting subject of turbulence modelling is too complex to enter into here from the theoretical point of view. However, it is worth mentioning two particular approaches since these form an important aspect of the guidelines given later.

The simplest approach to modelling the effect of turbulence is to assume that the combined effect of the Reynolds stresses mentioned above is as an additional viscosity, acting to produce fluid stresses which are simply the product of the (eddy) viscosity (ν_e) and the local velocity gradient. The calculation of this eddy viscosity can be approached in a number of ways, but the most commonly used method is that developed for the k-ε, two equation model in which:

$$\nu_e = C_\mu k^2 / \varepsilon \quad (15)$$

Where C_μ is a constant with a normally accepted value of 0.09, k is the turbulent kinetic energy per unit mass (that is the mean fluid kinetic energy associated with the fluctuating components of the velocity) and ε is the rate of dissipation of the turbulent kinetic energy per unit mass.

In the "standard" k-ε model, k and ε are solved for using transport equations for each quantity. These transport equations contain both classical advection and diffusion terms, but also "modelled" terms for production and dissipation. Their derivation is beyond the scope of this document. However, the key point to remember is that the k-ε model is generally applicable only to high Reynolds number flows with a turbulence structure that is homogenous, and in which production and dissipation of turbulence is in balance. The guidelines on this modelling approach (given later) list numerous cases for which these conditions do not apply and therefore where particular care may be needed.

It should also be noted that for problems in steady ship flows, this modelling approach is generally accepted to be unsatisfactory, other than for the most preliminary of assessments of the flow field.

An alternative to the above is to attempt to calculate each of the 6 Reynolds stresses directly through the solution of further transport equations for each component. These Reynolds Stress Transport methods are becoming accepted as feasible in application to ship hydrodynamics, and have been shown to give superior results to two equation modelling albeit at the cost of increased computing time. It should be emphasised that these too contain modelled terms, derived from a combination of theoretical argument and empiricism, and should be used with care.

Between the standard two equation modelling approach and solution of the RST equations there are a number of improvements and variations, such as RNG k-ε, k-ω models, and non-linear eddy viscosity models, all of which seek to overcome certain of the shortcomings of the standard k-ε model, without invoking too great a computational overhead. The suitability of such models for marine applications is problem dependent and will be addressed for each class of flow in the supplementary Application Procedures to be developed to accompany these general guidelines.

Finally, it should be recalled that all of the above discussion relating to turbulence modelling applies chiefly to flows with a steady mean. These modelling approaches may also be used where the mean flow varies over a time scale which is sufficiently large (slowly varying), or the eddies contained in the flow are sufficiently large, slow and weak, that the primary assumptions underlying the above equations are valid. It remains a matter of debate as to whether such assumptions are appropriate to hydrodynamic flows.

2.1.4. Potential Flow

Finally, it is possible to make further simplifications in order that a single scalar quantity, the fluid potential, can be used to describe the flow. If the flow is assumed inviscid and irrotational (i.e. potential flow) such that $\nabla \times \vec{v} = 0$ with $\nabla = (\partial/\partial x, \partial/\partial y, \partial/\partial z)$, the momentum equations reduce to the statement that fluid acceleration is directly related to the fluid pressure gradient. The Laplace equation for the fluid potential can be derived from continuity:

$$\nabla^2\phi = \partial^2\phi/\partial x^2 + \partial^2\phi/\partial y^2 + \partial^2\phi/\partial z^2 = 0 \quad (16)$$

Which is sufficient to determine the complete velocity field. As this equation is linear it is possible to combine elementary solutions, such as sources, sinks, doublets and vortices for application to complex solutions. It should also be appreciated that potential flow is the governing behaviour of gravitationally driven wave systems, and hence represents the fundamental physics, with appropriate boundary conditions, for free surface wave problems.

For potential flows, it is possible to derive a simple expression for the fluid pressure by integrating the Navier Stokes equations along a stream-line to give the well known Bernoulli equation.

Potential flows, and their characterisation using the Laplace equation, have many important and useful properties that can be used in the formulation of numerical solutions. The use of the Divergence Theorem by Gauss (to convert volume to surface integals), Green's theorem (to convert a surface integral to a line integral), and the principle of superposition of solutions, all provide the means to formulate boundary element solution methods. Various forms of boundary element or panel methods have, thus far, been the principal means by which these flows have been modelled. These methods are discussed in more detail below.

2.2. *General boundary conditions*

The numerical solution of the equations of fluid motion provided above, for any given hydrodynamic problem, require boundary conditions to be defined. These represent a unique description of the state of the flow at the geometrical boundaries of the three dimensional space within which the equations are to be modelled. There are in general, two types of boundary condition that can be applied, namely:

1. Where a fixed or prescribed value is defined for the variable of interest at known points on the boundary (the so called Dirichlet boundary condition)
2. Where the gradient (usually normal to the boundary) of the variable is known (the so-called Neumann condition)

Typical examples of the first kind can be found in the calculation of the flow field around a ship moving at constant forward speed, in an axis system moving with the vessel, and with the computational domain formed by a large control volume around the vessel within which the numerical solution is to be carried out. In this case, the fluid is assumed to enter the domain at an upstream boundary or inlet such that the ship appears stationary and the water flows past it. The inlet boundary velocity in this case is set to be a fixed value equal to the speed of the ship, and in the opposite direction. Similarly, on the ship surface, the values of the fluid velocity components are all set to zero (the so-called no-slip condition).

Examples of the second type of boundary condition can also be found in the numerical solution of steady ship flow problems. A symmetry plane is often assumed to lie along the ship's centreline that has the practical benefit of reducing the size of the computational domain. In cases for which the free surface effects are small or simply not of interest, the water-plane can also be assumed to be a symmetry plane (the so-called double-body problem). The symmetry boundary condition for the scalar pressure, and velocity components tangential to these boundaries, is that their gradients normal to these boundaries are zero.

The numerical implementation of these boundary conditions is dependent upon the type of solution method adopted. Guidance on their use is given later.

2.3. *Coupling with motions of floating systems*

2.3.1. *General comments*

The above discussion has centred upon problems associated with steady flows. For sea-keeping, manoeuvring, and the calculation of waves loads and responses of floating offshore platforms, the numerical solution of the fluid equations of motion require boundary conditions which reflect the dynamics of the problem.

There are two main areas of work to be considered. Problems which are characterised by regular harmonic solutions, or which can be developed from the superposition of harmonic solutions, are most often solved in the frequency domain. The linear sea-keeping problem, and certain types of motion of offshore platforms, are typical examples. Non-linear problems, on the other hand, are more open to the use of time domain simulation techniques.

2.3.2. *Linear Harmonic Non-steady Problems*

Strictly, the conditions required in order that frequency domain solutions can be applied are that the vessel or platform motions are small, that the boundary conditions are linear or can be linearized, and that the fluid equations of motion are of a form that allow principles of superposition to be used. Clearly this therefore falls within the realm of potential flow as discussed earlier, and in particular the field known as radiation and diffraction modelling.

The diffraction problem is that associated with the way in which the presence of a fixed or floating body distorts the pattern of ocean waves within which it sits, either through reflection or diffraction. The radiation problem is associated with the generation of waves by a floating body in response to the wave induced forces and moments acting on it and its subsequent dynamic response. There are 6 components to the radiated wave potential, each associated with a particular mode of vessel motion (surge, sway, heave, roll, pitch and yaw).

The flow and pressure fields within the fluid surrounding the vessel are calculated from the superposition of the incident wave field, the diffracted wave field and the radiated wave field. The total fluid potential is a complex quantity and is found from the complex summation (amplitude and phase) of the incident, diffracted and radiated components. The calculation of each of these components is made independently, with boundary conditions appropriate to each. For example, for the heave radiation problem at zero speed, the real and imaginary parts of the potential are calculated by solving a discrete, numerical, surface integral equation (derived by applying Gauss's Divergence theorem to the Laplace equation), over the wetted surface of the vessel. The boundary condition used is that which equates the vessel's heave velocity to the vertical component of the surface normal potential gradient. In practice, a unit amplitude of motion is used such that the vertical velocity is equal to the wave frequency of interest. Other components of motion are treated similarly.

2.3.3. *Non-Linear and Time Domain Simulations*

Non-linearities in hydrodynamic problems arise from a number of sources. Both steady and unsteady problems can exhibit sufficient non-linearity that simulation techniques are the only way to predict the flow and hydrodynamic pressure fields that result.

For problems in which the flow can be adequately described by a scalar potential, non-linearities can arise as the result of either large vessel motions (and hence changes in boundary surface area and or shape) or the need to apply non-linear forms of the free surface boundary conditions (discussed later). Nevertheless, the coupling of the fluid equations to the motions of the floating system remains as before, i.e. via the vessel surface velocity boundary condition. Since the free surface behaviour cannot be represented other than through non-linear time domain equations which describe its position (the kinematic condition) and pressure (the Bernoulli equation), the solution must be allowed to evolve by simulation.

For situations in which the RANSE equations are used to describe the fluid flow behaviour (e.g. where viscous effects are important), the problem is inherently non-linear and not open to the mathematical principles that allow the frequency domain approach to be used. Free surface motions and large-scale vessel motions are allowed also, and hence the solution techniques used are again those of time domain simulation.

The coupling of the vessel motion response with the solution of the hydrodynamic equations of motion requires that there is an explicit, parallel solution of the 6 degree of freedom rigid body equations of motion for the vessel. The hydrodynamic forces and moments acting on the vessel are calculated through and integration of pressures over its wetted surface at each step in the simulation. The resulting solutions for the vessels' motions are used to provide velocity components at the points required for the hull surface boundary condition.

2.4. Boundary element or panel methods for potential flow

We now move on from the general description of the fluid equations of motion to a discussion of the approaches used to solve firstly, potential flow problems using surface integral methods and secondly, RANSE methods in three dimensions.

Inviscid flow models remain the most important tool for studying offshore structures and remain the most reliable approach to wave resistance. They also provide the basis for the majority of propeller design methods. They all employ boundary integral formulations of various kinds and are therefore quite computationally efficient and, up until now, have offered the simplest approach to the modelling of free surface and propeller flows.

As an illustration of how these methods are developed, the particular case of the steady ship flow problem and the boundary integral formulation of its solution is described.

As noted earlier, the domain over which the flow solution is required is bounded by the wetted hull surface, the free surface, the sea bed (if sufficiently close), and a so-called far-field boundary. If the free surface height can be represented by $z = \zeta(x, y, t)$, the flow field is then evaluated by solving the Laplace equation everywhere for $z < \zeta(x, y, t)$:

$$\nabla^2\phi = 0 \tag{17}$$

where $= \nabla\phi$ is used to derive the flow velocities.

The following boundary conditions are formulated throughout the domain:

Kinematic boundary conditions: Water does not penetrate the free surface or the body surface.

Dynamic free-surface boundary condition: Atmospheric pressure acts at the water surface, which is considered to contain all surface streamlines. This allows the use of the Bernoulli equation in the formulation of a condition for the unsteady potential in combination with the kinematic condition mentioned above.

Radiation or far field boundary conditions: Which depend on the type of analysis undertaken, but can be summarised as allowing the propagation of waves in the far field which satisfy the need for consistency in the transport of energy away from the disturbance. For linear wave resistance or radiation / diffraction problems, these conditions are implicit in the choice of Green's function (see below). For non-linear time domain or field methods, the computational domain is truncated at some distance from the vessel and appropriate numerical models that satisfy the required properties are applied.

The steady kinematic condition on the water surface $z = \zeta$ can be written:

$$\nabla\phi \cdot \nabla\zeta = \phi_z \tag{18}$$

The steady dynamic condition at $z = \zeta$ is:

$$gz + \frac{1}{2}(\nabla\phi)^2 = \frac{1}{2}U^2 \tag{19}$$

The non-linear free surface boundary condition for is formed by combining the kinematic and dynamic boundary conditions:

$$\frac{1}{2}\nabla\phi \cdot \nabla(\nabla\phi)^2 + g\phi_z = 0 \tag{20}$$

It can be assumed that the total potential is made up of a free-stream potential and a smaller perturbation potential. The linearised Kelvin free-surface boundary condition at the undisturbed surface is then: