



BEST PRACTICE
GUIDELINES
FOR
MARINE APPLICATIONS
OF COMPUTATIONAL
FLUID DYNAMICS

*Prepared by
WS Atkins Consultants
And members of the NSC*

*Sirehna
HSVA
FLOWTECH
VTT*

*Imperial College of Science & Technology
Germanischer Lloyd
Astilleros Espanoles*

MARNETCPD

MARCI
5/10/03
B

[HTTPS://PRONET.WSATKINS.CO.UK/MARNET/PUBLICATIONS/BPG.PDF](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 led 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 led 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:

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

Momentum equations:

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

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

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

Energy equation:

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

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:

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

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

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

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:

$$\nabla \cdot U = 0 \quad (8)$$

The momentum equations become:

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

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

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

Where D/Dt is the substantial derivative given by:

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

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:

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

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:

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

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 $k-\varepsilon$, 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" $k-\varepsilon$ 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 $k-\varepsilon$ 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 $k-\varepsilon$, $k-\omega$ models, and non-linear eddy viscosity models, all of which seek to overcome certain of the shortcomings of the standard $k-\varepsilon$ 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 integrals), 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 an 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 $z < \zeta(x, y, t)$:

$$\nabla^2 \phi = 0 \quad (17)$$

where $\mathbf{v} = \nabla \phi$ 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:

$$\nabla \phi \cdot \nabla \zeta = \phi_z \quad (18)$$

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

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

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

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

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:

$$\phi_{xx} = \frac{g}{U^2} \phi_z \quad (21)$$

The wave resistance can be calculated from the energy in the free wave spectrum, or by integration of the steady hull surface pressures arising from the solution.

In a higher order approach suggested by Dawson [1977], the total potential is divided into a double-body potential and a perturbation potential. As the perturbation potential is small, the double-body potential corresponds to the limiting solution as the Froude number goes to zero. More advanced methods have since been developed, such as Raven [1996] and Janson [1997] although Dawson's method remain the basis of many computer codes. A fully non-linear approach is also possible whereby both the kinematic and dynamic boundary conditions are satisfied by an iteration procedure that has many similarities with time domain simulation.

In order to solve for the fluid potential, the Laplace equation is transformed into a surface integral taken over the ship hull as described earlier. This surface integral then provides the means to develop a discrete set of integral equations by splitting the surface up into a number of panels. One integral equation is then written for each panel in which the hull surface boundary condition is satisfied locally.

Each panel is assumed to represent a fluid source, which may be a local point value, or may be distributed in some pre-defined manner (i.e. constant strength per unit area, bi-linear distribution, etc.).

Each panel source has an effect on every other panel source, contributing to the induced flow over each panel surface. The influence of each panel on every other panel is represented by a weighting function, which in classical hydrodynamics is known as a Green's function.

The Rankine source is used for steady flow in an infinite fluid and is one of the simplest functions used. It simply makes the assumption that the velocity potential induced at a field point some distance from the source is inversely proportional to the distance between them. This type of function is suitable for both linear and non-linear applications.

More complex functions are used for linear free surface flows with wave radiation. The main classes of such functions are:

- The zero speed pulsating source Green's function,
- The steady forward speed Green's function,
- The translating, pulsating source Green's function,

For linear problems, these functions provide weightings for the source potential which also uniquely satisfy the linear free surface and far-field boundary conditions, removing the need to distribute further sources over these regions, and greatly reducing computing times.

For each panel therefore, the normal fluid velocity induced over the panel surface is calculated from the summation of all other source (or dipole) contributions weighted by a Green's function, and its own self-influence. This velocity is then equated to the boundary condition at the panel surface, which requires no net flow through the panel. In the case of a general steady flow, with a non-linear free surface boundary condition to be satisfied, and in the presence of lifting surfaces, this is expressed in the pair of equations:

$$\phi = \iint_{S_B} \sigma_B G_B .dS + \iint_{S_F} \sigma_F G_F .dS + \iint_{S_W} \mu_W G_W .dS \quad (22)$$

$$(U + \nabla \phi) \cdot \vec{n} = 0 \quad (\text{on the hull}) \quad (23)$$

Where subscript B denotes the surface of the body, F the free surface and W a trailing wake sheet comprising a dipole distribution, as applicable. Here σ is the element source strength, μ the dipole strength, and G the corresponding Green's function.

Equation (6) may be differentiated and substituted into (7) to provide allow the numerical discretisation of the surface integral. This can then be expressed in matrix form as:

$$[D_{ij}] \phi + [W_{ik}] \nabla \phi = [S_{ij}] (U_{\infty} \cdot n) \quad (24)$$

where for panel j , S_{ij} is the source influence coefficient of a unit strength panel, D_{ij} is the dipole influence coefficient and W_{ik} is the influence of the constant strength wake strip extending to infinity.

The overall solution is achieved by inversion of this matrix problem using standard techniques, and from the subsequent recovery of potentials, source and dipole strengths.

2.5. Methods for viscous turbulent flows

2.5.1. General

The differences between viscous turbulent flow solvers and the previously described potential flow methods are numerous. They stem from the considerably more complex forms of non-linear partial differential equation being addressed, and the need to carry out the numerical discretisation in 3D, rather than being able to reduce the problem to a set of surface integrals.

The principle which is common to all is that the fluid computational domain is split into a three dimensional grid of data points. This grid may be "structured" or "unstructured" depending upon the details of the numerical scheme and solvers employed.

Structured grids represent the simplest type and were used in the earliest forms of numerical solution schemes. Such grids contain fixed distributions of grid points in all principal co-ordinate directions. This is made less restrictive by the use of numerical mapping schemes that allow the generation of so-called body fitted meshes to fit complex curved surfaces at domain boundaries. However, the overall shape of the computational domain must be essentially 6 sided (in a Cartesian co-ordinate system). Some of the difficulties arising from this restriction can be overcome by use of multi-block techniques. This method allows more complex volume geometries to be generated by joining large numbers of hexahedral blocks together. This so-called structured multi-block method is probably the most commonly used approach in marine applications of CFD at the present time.

Unstructured meshes have no requirement for such consistency. Computational domains can be of arbitrary shape and can be discontinuous, so long as the grid volumes used to fill the space tessellate, so as to leave no gaps or disconnection between volumes. In the majority of CFD formulations that use unstructured grids, a variety of grid volume shapes can be employed, e.g. hexahedra, tetrahedra, prisms, etc. This is rapidly becoming the most common approach in general applications of CFD in industry, owing primarily to the development of tools for automatic grid generation.

The following represent the main types of approach used in the formulation of the RANS Equations for numerical solution on both types of grid.

2.5.2. Finite difference method

The finite difference method is the oldest of the methods, considered to have been developed by Euler in 1768, and is used to obtain numerical solutions to differential equations by hand calculation. At each node point of the grid used to describe the fluid domain, Taylor series expansions are used to generate finite difference approximations to the derivatives of the RANS equations. The derivatives appearing in the governing equations are then replaced by these finite difference expressions, yielding an algebraic equation for the flow solution at each grid point.

It is the simplest method to apply, but requires a high degree of regularity of the mesh. In general, the mesh must be structured. Grid points should form an ordered array in three dimensions, allowing the finite difference approximations to be formed from local, easily addressed locations. Grid spacing need not be uniform, but there are limits (guidelines given later) on the amount of grid stretching or distortion that is possible, and at the same time maintain accuracy. Topologically, these finite difference structured grids must fit the constraints of general co-ordinate systems with, for example, Cartesian grids must fit within 6 sided computational domains. However the use of an intermediate co-ordinate mapping allows this otherwise quite major geometrical constraint to be relaxed, such that complex shapes (including ship hulls) can be modelled.

2.5.3. Finite element method

The finite element method was developed initially as a procedure for constructing matrix solutions to stress and displacement calculations in structural analysis. The method uses simple piecewise polynomial functions on local elements to describe the variations of the unknown flow variables. When these approximate functions are substituted into the governing equation it will not hold exactly, and the concept of a residual is introduced to measure the errors. These residuals are then minimised by multiplying by a set of weighting functions and then integrating. This results in a set of algebraic equations for the unknown terms of the approximating functions and hence the flow solution can be found.

Finite element methods are not used extensively in CFD, although there are a number of commercial and research based codes available. For certain classes of flow, FE methods bring a high degree of formalised accuracy to the numerical modelling process. However, it has generally been found that FE methods require greater computational resources and cpu effort than equivalent Finite Volume methods, and therefore their popularity, at least in Europe, is limited.

2.5.4. Spectral method

Spectral methods use the same general approach as the finite difference and finite element methods by again replacing the unknowns of the governing equation with truncated series. The difference is that, where the previous two methods use local approximations, the spectral method approximation is valid throughout the entire domain. The approximation is either by means of truncated Fourier series or by series of Chebyshev polynomials. The discrepancy between the exact solution and the approximation is dealt with using a weighted residuals concept similar to finite element.

2.5.5. Finite volume method

The finite volume method was first introduced by McDonald [1971] and MacCormack and Paullay [1972] for the solution of two dimensional time dependent Euler equations, and extended to three dimensional flows by Rizzi and Inouye [1973]. The method discretises the integral form of the conservation laws directly in physical space. The resulting statements express the exact conservation of relevant properties for each finite cell volume. Finite-difference-type approximations are then substituted for the terms of the integrated equations, forming algebraic equations that are solved by an iterative method.

As the method works with the cell volumes and not the grid intersection points, unstructured meshes can be used where a large number of options are open for the definition of the shape and location of the control volumes around which the conservation laws are expressed. A 'finite element' type mesh can be used where the mesh is formed by combinations of triangular or quadrilateral cells (or tetrahedra and pyramids in three dimensions), where the mesh cannot be identified with co-ordinates lines. This type of unstructured mesh, although requiring careful bookkeeping, can offer greater flexibility for complicated geometries.

Flow variables can be stored either at Cell Centre or Cell Vertex locations. Conveniently, the cells coincide with the control volumes if using the Cell Centred scheme. For the Cell Vertex scheme, additional volumes are required to be constructed, however, the scheme has the



advantage that boundary conditions are more easily applied since the variables are known on all boundaries.

Finally, it should be noted that, of all the methods described above, the Finite Volume method is by far the most common approach to be found in current commercial CFD codes. Much of the guidance given in this document is based on the assumption that the reader is following this approach.



2.6. Dealing with the free surface

2.6.1. Potential flow

The primary difficulty with free surface calculations is that the position and shape of the free surface is not known, and often involves non-linear effects such as wave breaking and fragmentation. In any case, wave diffraction and radiation effects can be substantial for many marine structures with large dimensions.

Earlier, some background description of free surface flow boundary conditions was given, dealing with both the linear frequency domain and the linearized steady ship wave problem. In both cases, solutions are achievable using a suitable Green's function on the hull which explicitly satisfies the linear free surface boundary condition, thereby removing the need to model further the behaviour of the free surface.

However, the full definition of the free surface boundary condition is both non-linear as a mathematical statement and in its geometrical location. Recalling equations 2 and 3 given earlier, and considering their unsteady or dynamic form, we get:

The unsteady kinematic condition,

$$\frac{\partial \zeta}{\partial t} = \phi_z - \nabla \phi \cdot \nabla \zeta \quad (25)$$

and the unsteady dynamic condition:

$$\frac{\partial \phi}{\partial t} + gz + \frac{1}{2}(\nabla \phi)^2 = \frac{1}{2}U^2 \quad (26)$$

These coupled equations are posed on the free surface itself, and therefore to be considered within a moving frame of reference. Note that here, the potential considered is the total fluid potential.

For non-linear, steady or unsteady problems solved using a panel method, one method of approach is to simulate the evolution of a steady state, or transient behaviour of the flow by discretising the above equations in time as well as space. For example the first order time derivative in the above equations can be replaced with a simple forward finite difference expression or by more advanced Runge-Kutta like time marching schemes. In these cases, the kinematic condition is used to update the location of the free-surface panels used to describe the boundary at each time step, usually using the values of velocity potential and surface elevations at the current step. The formulation of the boundary condition for the potential using the unsteady dynamic condition is more complex. The simplest approach is clearly to update the potential, and use this predicted value in the boundary integral formulation. However, other methods which seek to couple the solution of the free surface potential to that on the surface of the vessel directly have also been developed but are beyond the scope of this document to describe in detail.

It should also be noted that these non-linear potential flow problems are still open to the use of the principle of superposition of solutions. For sea-keeping or offshore engineering problems, it is therefore common to assume a single frequency harmonic incident potential, and to express the problem in terms of the non-linear perturbation or solution potential accordingly.

2.6.2. Viscous flow

2.6.2.1. Interface tracking

There are essentially two approaches to free surface modelling for viscous flows using RANSE solvers: interface tracking and interface capturing.

Interface tracking involves generation of a grid covering just the liquid domain. One of the domain boundaries is then, by default, the free surface where the boundary conditions are applied. The grid is adapted to the position of the free surface at each time step. Grid adaptation may be made computationally more efficient by methods such as moving points along predefined lines or by updating the free surface position only after several time steps, having solved the free surface using the pressure boundary condition at intermediate steps. The method can currently only be used in the absence of steep or breaking waves to avoid contortion of the grid. Use of unstructured meshes may improve these limitations.

2.6.2.2. Interface capturing

The alternative approach, interface capturing, involves solving the RANS equations on a predetermined grid which covers the whole domain. Three main methods cover this category:

- Marker-and-cell: Massless tracer particles are introduced into the fluid near the free surface and tracked throughout the calculation. This scheme can cope with non-linearities such as breaking waves and has produced some good results. However, it is computationally expensive.
- Volume of fluid (VOF): The two fluid phases are considered to make up one single fluid. The position of each phase is described by assigning a volume fraction of either 0 or 1. The free surface is then identified with the region of rapid change in this volume fraction. The volume fraction is solved for one of the phases by means of an extra transport equation, having generally the same form as the mass transport equation. One common algorithm used for this solution is SIMPLE. A pressure-correction equation is obtained from the discretised form of the mass and momentum equations. An initial estimation of the velocity components is made from the momentum equations, which are then corrected by solving the pressure-correction equation. This enables the solution of the equations for volume fraction, turbulent kinetic energy, energy dissipation rate and eddy viscosity. The equations are solved iteratively until within the set tolerance. Interface sharpening algorithms are commonly used to refine those cells with a value of between 0 and 1. Recently, advances have been made in modelling ship motions in a seaway using VOF methods. Motions are forced by imposing an oscillatory motion to the hull. As this takes into account the viscosity it is maybe not surprising that the results for effects such as roll damping coefficient are better than from the inviscid methods, although such methods are still in their infancy.

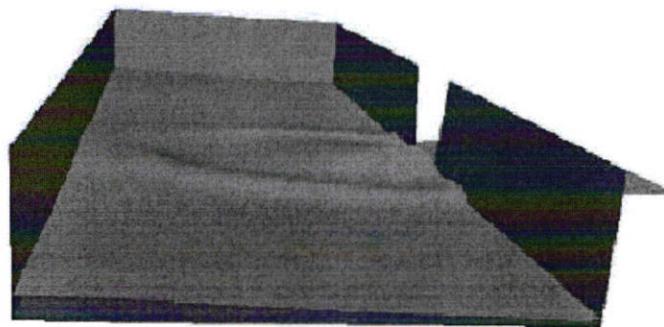


Figure 2.1 VOF method used to study water ingress on a damaged RoRo deck

- Level set technique: A scalar "level set" function is defined in each cell. Initially, it is set equal to the distance from the free surface, positive in one direction and negative in the other. At every later instance the function is computed from the condition that its total (material) derivative with respect to time is zero. This means that the value of the function is constant with time on all fluid particles on the free surface. These points will always be on the free surface, since the relative normal velocity is zero. Thus the surface can be found at each time by finding the surface of zero value of the level set function. The surface is obtained in both air and water, but a smoothing layer needs to be introduced at the interface where the density and viscosity exhibit large jumps.

The main disadvantage of interface capturing over interface tracking is the need to predict where grid refinement is required as the location of breaking waves, etc. will not generally be known in advance.

3. General errors and uncertainties in CFD simulations

3.1. Sources of errors and uncertainties and their classification

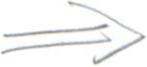
Within these guidelines the term Computational Fluid Dynamics (CFD) is commonly used to describe a variety of techniques used to solve fluids engineering problems. These techniques range from panel methods used to solve potential flow problems to finite volume techniques used to examine fully turbulent flows. Although the underlying physical equations and the solution techniques vary, they commonly involve the replacement of the governing equations with a discrete representation and the numerical solution of these approximate equations using a computer. This discretisation process means that in all cases the solutions obtained are approximate. Furthermore, fluid flow processes are physically complex and in certain cases the governing equations are only an approximate representation of the true physical processes. A typical example of this sort of uncertainty occurs with the use of a turbulence model when performing a viscous flow simulation.

In addition to the source of errors and uncertainty that are introduced by the numerical model, the CFD engineer can also introduce errors and uncertainties. The process of performing a CFD calculation is itself complex and requires the engineer to perform a number of different activities. These typically include:

- 
- definition of the problem;
 - selection of the solution strategy;
 - development of the computational model;
 - analysis and interpretation of the results.

All of these steps are potentially error prone or subject to some degree of uncertainty.

There is no universally accepted means of identifying or classifying errors, which can range from human or user errors to inadequacies in the modelling strategy and model equations. However, the ERCOFTAC BPG adopts the following classification based on seven different sources of error and uncertainty:

- 
1. Model error and uncertainties;
 2. Discretisation or numerical error;
 3. Iteration or convergence error;
 4. Round-off error;
 5. Application uncertainties;
 6. User errors;
 7. Code errors.

This categorisation has been adopted for these guidelines and, in common with the ERCOFTAC BPG, is used to structure the guidance. For the present purpose, however, which considers both potential and viscous flow calculations, guidance is presented in three sections. The first considers the sources of errors and uncertainties that are common to both solution methods and a further two that focus on advice which is relevant to the particular methods.

3.1.1. Model error and uncertainty

These are defined as errors due to the difference between the real flow and the exact solution of the model equations. This includes errors due to the fact that the exact governing flow equations are not solved but are replaced with a physical model of the flow that may not be a

good model of reality. For viscous simulations, the most well publicised error in this category is the error from turbulence model and for potential flow calculation viscous effects are neglected altogether.

In short, the model errors and uncertainties can be described as the errors that arise because we are in fact solving the wrong equations.

3.1.2. Discretisation or numerical error

These are defined as errors that arise due to the difference between the exact solution of the modelled equations and a numerical solution on a grid with a finite number of grid points. In general, the greater the number of grid cells, the closer the results will be to the exact solution of the modelled equations, but both the fineness and the distribution of the grid points affect the result. This type of error arises in all numerical methods and is related to the approximation of a continually varying parameter in space by some polynomial function for the variation across a grid cell. In first order schemes, for example, the parameter is taken as constant across the cell. In short, discretisation errors arise because we do not find an exact solution to the equations we are trying to solve but a numerical approximation to this.

3.1.3. Iteration or convergence error

These are defined as errors which arise due to the difference between a fully converged solution on a finite number of grid points and a solution that is not fully converged. The equations solved by CFD methods are generally iterative, and starting from an initial approximation to the flow solution, iterate to a final result. This should ideally satisfy the imposed boundary conditions and the equations in each grid cell and globally over the whole domain, but if the iterative process is incomplete then errors arise. In short, convergence errors arise because we are impatient or short of time or the numerical methods are inadequate and do not allow the solution algorithm to complete its progress to the final converged solution.

3.1.4. Round-off errors

These are defined as errors that arise due to the fact that the difference between two values of a parameter during some iterative scheme is below the machine accuracy of the computer. This is caused by the limited number of computer digits available for storage of a given physical value.

3.1.5. Application uncertainties

Inaccuracy that arises because the application is complex and precise data needed for the simulation is not available. Examples of this are uncertainties in the precise geometry, uncertain data that needs to be specified as boundary conditions and uncertainties as to whether the flow is likely to be steady or unsteady.

3.1.6. User Errors

These are defined as errors that arise due to mistakes and carelessness of the user. Such errors generally decrease with increasing experience of the user, but in the nature of things cannot be completely eliminated as "to err is human". This error is often described by the popular jibe "garbage in, garbage out".

3.1.7. Code Errors

These are the errors due to bugs in the software, unintended programming errors in the implementation of models or compiler errors on the computer hardware being used. Such errors are often difficult to find, as CFD software is highly complex, typically involving hundreds of thousands of lines of code for a commercial product. Computers are very unforgiving. Even a relatively simple typing error that might easily be overlooked on this page,

such as an "i" for a "j" in a single word, when incorporated into a single line of code, can have disastrous consequences.

3.2. Definitions of errors and uncertainties

The deficiencies or inaccuracies of CFD simulations can be related to a wide variety of errors and uncertainties. A recent publication of the AIAA guide for the verification and validation of computational fluid dynamics simulations [1998] provides useful definitions of error and uncertainty in CFD as follows:

Error: A recognisable deficiency that is not due to lack of knowledge.

Uncertainty: A potential deficiency that is due to lack of knowledge.

These rather philosophical definitions can be made clearer by examples. Typical known errors are the round-off errors in a digital computer and the convergence error in an iterative numerical scheme. In these cases, the CFD analyst has a reasonable chance of estimating the likely magnitude of the error. Unacknowledged errors include mistakes and blunders, either in the input data or in the implementation of the code itself, and there are no methods to estimate their magnitude. Uncertainties arise because of incomplete knowledge of a physical characteristic, such as the turbulence structure at the inlet to a flow domain or because there is uncertainty in the validity of a particular flow model being used. An error is something that can be removed with appropriate care, effort and resources, whereas an uncertainty cannot be removed as it is rooted in lack of knowledge.

3.3. Definitions of verification, validation and calibration

In discussions of CFD errors and uncertainties it is useful to make some clear distinctions between the meaning of the terms validation, verification and calibration. The definitions used in these guidelines follows closely the similar definitions given in the AIAA guide [1998] Roache [1998], Rizzi and Vos [1998] and Fisher and Rhodes [1996]:

Verification: Procedure to ensure that the program solves the equations correctly.

Validation: Procedure to test the extent to which the model accurately represents reality.

Calibration: Procedure to assess the ability of a CFD code to predict global quantities of interest for specific geometries of engineering design interest.

In the field of ship hydrodynamics, much work has been undertaken on this subject and published in the ITTC reports.

4. Method independent errors and uncertainties - Guidelines

4.1. Convergence errors

Iterative algorithms are used for steady state solution methods and for procedures to obtain an accurate intermediate solution at a given time step in transient methods. Progressively better estimates of the solution are generated as the iteration count proceeds.

There are no universally accepted criteria for judging the final convergence of a simulation, and mathematicians have found no formal proof that a converged solution for the Navier-Stokes equations exists. In some situations the iterative procedure does not converge, but either diverges or remains at a fixed and unacceptable level of error, or oscillates between alternative solutions. Careful selection and optimisation of control parameters (such as damping and relaxation factors or time-steps) may be needed in these cases to ensure that a converged solution can be found.

The level of convergence is most commonly evaluated based on residuals, on values of globally integrated parameters, such as lift coefficient or heat transfer coefficient, or on time/iteration signals of a physical quantity at a monitor point, which is an arbitrarily chosen location in the flow domain.

Residuals

Residuals are 3D fields associated with a conservation law, such as conservation of mass or momentum. They indicate how far the present approximate solution is away from perfect conservation (balance of fluxes). Usually, the residuals are normalised by dividing by a reference value, which may be one of the following:

- Maximum value of the related conserved quantity.
- Average value of the related conserved quantity.
- Inlet flow of a related quantity.

Convergence is usually monitored on the basis of one representative number characterising the residual level in the 3D flow field. This single value may be:

- A maximum value.
- The sum of absolute values.
- The sum of squared values.
- The arithmetical average of absolute values.
- The root-mean-square value.

The large number of variants makes it difficult to give precise statements how to judge convergence and at which residual level a solution may be considered converged. In principle, a solution is converged if the level of round-off error is reached. Special care is needed in defining equivalent levels of convergence if different codes are used for comparison purpose.

Recommendations to the code developers:

- CFD codes should make available the maximum possible information to judge convergence. This includes residuals for every conserved quantity.
- Give information on the spatial distribution of residuals.
- Residuals should be dimensionless.
- Clear definition in the handbook how the residuals are determined.
- To avoid confusion of the CFD users, one commonly accepted definition of the residual should be adopted.

Guidelines

- Be aware that different codes have different definitions of residuals.
- Always check the convergence on global balances (conservation of mass, momentum and turbulent kinetic energy) where possible, such as the mass flow balance at inlet and outlet and at intermediate planes within the flow domain.
- Check not only the residual itself but also the rate of change of the residual with increasing iteration count.
- Convergence of a simulation should not be assessed purely in terms of the achievement of a particular level of residual error. Carefully define solution sensitive target quantities for the integrated global parameters of interest and select an acceptable level of convergence based on the rate of change of these (such as mass flow, lift, drag, and moment forces on a body).
- For each class of problem carry out a test of the effect of converging to different levels of residual on the integrated parameter of interest (this can be a single calculation that is stopped and restarted at different residual levels). This test demonstrates at what level of residual the parameter of interest can be considered to have converged and identifies the level of residual that should be aimed at in similar simulations of this class of problem.
- Monitor the solution in at least one point in a sensitive area to see if the region has reached convergence.
- For calculations that are proving difficult to converge, then the following advice may be helpful:
 - Use more robust numerical schemes during the first (transient) period of convergence and switch to more accurate numerical schemes as the convergence improves.
 - Reduce parameters controlling convergence, for instance under relaxation parameters or the CFL number.
 - If the solution is heavily under-relaxed increase relaxation factors at the end to see if the solution holds.
 - Check whether switching from a steady to a time-accurate calculation has any effect.
 - Consider using a different initial condition for the calculation.
 - Check the numerical and physical suitability of boundary conditions (see also Section 3.7.3 and Chapter 5)
 - Check whether the grid quality in areas with large residual has any effect on the convergence rate.
 - Look at the residual distribution and associated flow field for possible hints, e.g. regions with large residuals or unrealistic velocity levels.

4.2. Round-off errors

Round-off errors are not usually of great significance. But in situations where the small arithmetical differences between two large numbers become relevant, cancellation due to round-off may lead to severe errors. To avoid large values it is common practice to calculate pressure relative to a reference value. Examples where round-off errors are known to be of significance are:

- Low Reynolds number turbulence models with large exponential terms.
- Flows with density driven buoyant forces with small density and temperature differences.
- High aspect ratio grids with large area ratios on different sides of the grid.
- Conjugate heat transfer.
- Calculations of scalar diffusion with low concentrations of one species.
- Low Mach number flows with a density based solver.
- Flows with large hydrostatic pressure gradients.

Guidelines

- Always use the 64-bit representation of real numbers (double precision on common UNIX workstations).
- Developers are recommended to use the 64-bit representation of real numbers (REAL*8 in FORTRAN) as the default settings for their CFD code.

4.3. Spatial discretisation errors

Different numerical methods evaluate the fluxes at the same grid locations as the transported quantities or somewhere in between (collocated or staggered grids). In both cases, an algebraic approximation of the spatial functions is required to calculate the gradients at these locations. This approximation is called the differencing scheme in finite volume or difference methods or the basis function in finite element methods. The accuracy of the scheme depends on the form of the algebraic relationship and on the number of grid points used in it (stencil). The spatial discretisation or truncation error equals the difference between the scheme and the exact formulation based on a Taylor expansion series. A formally second order scheme is consistent with the exact formulation up to the term with a power of two, a third order scheme also takes into account the next higher term. The formal order of accuracy is not preserved on irregular meshes, where it reduces by one. Reducing the cell size by introducing a finer grid has the biggest impact on the accuracy of the solution if higher order schemes are applied. Halving the elements in all directions using a 3rd order scheme will reduce the numerical error by a factor of 8, while this factor is only 2 with a 1st order scheme.

If the solution of the physical problem considered is smooth and exhibits only small gradients even a first order scheme can do a good job, but it is not at all suitable for general engineering applications involving complex flows with large gradients and thin boundary layers. The large truncation error introduced by the first order upwind scheme, particularly popular in finite volume methods, is known as numerical viscosity or diffusivity as it gives rise to artificial diffusion fluxes, which may be much stronger than the real molecular or turbulent contributions.

On the other hand higher order schemes suffer from a different more obvious problem, namely the appearance of a characteristic wavy pattern with a wavelength of two cell sizes in the neighbourhood of steep gradients. These so-called wiggles are caused by dispersion errors, i.e. waves with different wave lengths are not transported with the same speed. Dispersion errors are most prominent in central differencing schemes for finite volume methods and quadratic basis function schemes for finite element methods. Higher order upwind schemes are less prone to it. If necessary, this problem may be remedied using special (non-linear) TVD or shock-capturing schemes. Due to their capability to resolve steep gradients or interfaces while avoiding dispersion effects they are frequently applied in supersonic flows with shock waves or for the transport of scalar quantities with weak molecular diffusion.

Guidelines

- Avoid the use of 1st order upwind schemes. The use of methods of higher order (at least 2nd) is recommended for all transported quantities. It may be necessary to use a 1st order scheme at the start of a calculation as it is likely to be more robust, but as convergence is approached a 2nd order or higher scheme should be used.
- Try to give an approximation of the numerical error in the simulation by applying a mesh refinement study or if this is not possible by mesh coarsening.
- If available in the code, make use of the calculation of an error estimator (which may be based on residuals or on the difference between two solutions of different order of accuracy).

4.4. Temporal discretisation for unsteady problems

Purely steady flow fields with the time-derivative equal zero are only a special case of the time-dependent equations. In general, fluid flows are transient, whereby the sources for this time-dependent behaviour are:

- External transient or non-transient forces.
- Transient boundary conditions, moving walls (e.g. the fluttering of an airfoil).
- Vortex stretching, a three-dimensional phenomenon due to the non-linear term of the governing equations, which also gives rise to the fluctuating nature of turbulence.

The computation of steady turbulent flow is the most common kind of simulation for the general use of CFD. In these cases the Reynolds-averaged flow is steady while the average turbulent quantities account for the time-dependence of the turbulent fluctuations. However, the RANS-equations also allow the time-dependent Reynolds-averaged flow fields to be computed, based on the assumption that the temporal average of the turbulent quantities is not affected by the global unsteadiness. This is physically correct if the spatial scale of the turbulent eddies is much smaller than the geometrical scale of the analysed geometry. A time-dependent simulation is always needed if the scale of eddies or vortices becomes larger and is in comparable size to the dimensions of the geometry (e.g. the computation of vortex shedding).

If an accurate spatial discretisation is applied, flows which are physically time-dependent will fail to converge using a steady-state method. Very often convergence problems with a steady simulation can be interpreted as a hint that the flow is unsteady and a time-stepping scheme would be appropriate. On the other hand, symmetry boundary conditions may impose a steady flow, although it would be transient in reality. If the complete geometry including both sides of the symmetry plane were used the velocity field would oscillate perpetually. Averaging the solution over a long time interval would lead to a symmetrical field, which, however, differs from the steady state solution with the symmetry plane.

The temporal discretisation scheme provides an approximation of the time derivative. Most CFD codes offer first order and second order schemes, which are unconditionally stable and most effective in terms of computer memory and stability requirements. Low-storage higher-order Runge-Kutta methods are also available. The order of the scheme and the choice of the time step influence the size of the amplitude and the phase error, the two components of the temporal discretisation error. To improve time-accuracy self-adaptive time-stepping procedures (such as predictor-corrector methods) can be used.

The choice of the time step depends on the time scales of the flow being analysed. If time steps are too large the simulation might fail to capture important flow and mimic unphysical steady behaviour. It is therefore advisable to start with relatively small CFL numbers¹ even though this is not required from the point of view of numerical stability. Some CFD codes use a time stepping scheme for steady state simulations. It should be noticed that the accuracy of the converged steady state result is not completely independent of the time step. Special care is required to avoid choosing a time step which is too large.

Guidelines

- The overall solution accuracy is determined by the lower order component of the discretisation. At least second order accuracy is recommended in space and time. For time dependent flows the time and space discretisation errors are strongly coupled. Hence finer grids or higher order schemes are required (in both space and time).
- Check the influence of the order of the temporal discretisation by analysis of the frequency and time-development of a quantity of interest (e.g. the velocity in the main flow direction).
- Check the influence of the time-step on the results.

¹ The CFL number for incompressible flow is defined as $CFL = \Delta t v / \Delta x$, where Δt is the time step, Δx the local cell size and v the local velocity.

- Ensure that the time-step is adapted to the choice of the grid and the requested temporal size by resolving the frequency of the realistic flow and ensure that it complies with eventual stability requirements.

4.5. Geometrical uncertainties

In many industrial and engineering problems, the geometry of the object to be simulated is extremely complex and requires much effort to specify it exactly for a computer simulation. There are many sources of error which can arise in this process, such as:

- Changes in geometry that have occurred during the design process have been neglected.
- CAD geometry definition is insufficiently complete for flow simulation. Some surfaces and curves may not meet at the intended end point locations due to different levels of accuracy in different parts of the CAD model. Other curves may be duplicated.
- The geometry of a tested component may get modified during the testing procedure, and these modifications may not have been added to the original drawings.
- The geometry may not be manufactured within the tolerances as shown on the drawing, particularly with regard to fine flow features, such as the rounded shape of propeller leading edges, or symmetrical features.
- The effective geometry of the surface may have changed during use due to wear, erosion or fouling, such as marine growth.
- Small details of the geometry may have been omitted, such as roughness on the walls, welding fillet radii, small protrusions from the body, etc.
- The co-ordinate system used in the CAD system may be different from that used in the CFD code (rotational direction).

Guidelines

- Check and document that the geometry of the object being calculated is the geometry as intended. For example, the transfer of geometrical data from a CAD system to a CFD system may involve loss of surface representation accuracy. Visual display of the geometry helps here.
- In general, it is not necessary to explicitly include geometrical features that have dimensions below that of the local grid size provided that they are taken into account in the modelling (e.g. roughness in wall layer).
- In areas where local detail is needed then grid refinement in local areas with fine details should be used, such as in the neighbourhood of fine edges, or small clearance gaps. If grid refinement is used the additional grid points should lie on the original geometry and not simply be a linear interpolation of more grid points on the coarse grid.
- Check that the geometry is defined in the correct co-ordinate system and with the correct units which are requested by the CFD-code. CAD-systems often define the geometry in millimetres and this must be converted to SI-units if the code assumes that the geometry information is in these units. This is commonly done by most codes.
- If the geometry is altered or deformed by the hydrodynamic, mechanical or thermal loading, then some structural/mechanical calculation may be necessary to determine the exact geometry.

4.6. User errors

4.6.1. General comments

In CFD the human factor plays an important role, as the results depend to a large extent on the competence and expertise of the user. It is worthwhile spending a few words on this rather embarrassing aspect of CFD, as it is one of the prime causes of uncertainty in the results of CFD simulations. This may help to avoid some, if not all, of the most easily avoidable mistakes in the future. Several factors may give rise to user errors:

- Lack of attention to detail, sloppiness, carelessness, mistakes and blunders.
- Too optimistic and uncritical use of CFD, thanks to the high accessibility through simple interactive graphical user interfaces in commercial software, and the convincing and seductive power of the colourful visualisations.
- Lack of experience so that the user is unaware of a technical difficulty or unaware that critical information is missing.
- Unfamiliarity with a particular CFD code, and the tacit assumption that certain parameter settings are equivalent to those in a code with which the user is more familiar.

While the first two points are associated with the user's attitude and personal disposition the remaining points refer to the question of experience and training.

4.6.2. Control of the working process

Many mistakes are made by mere lack of attention to detail, or because the user is not aware of factors that can give rise to them. The best way to deal with these issues is for the user to have a clear checklist of issues that can arise which helps to ensure that all relevant problem areas have been dealt with. This becomes most important if the user has limited experience.

A formal management Quality Assurance (QA) system with checklists can help to support the inexperienced user to produce quality CFD simulations. It has been noted by Roache [1998], however, that a CFD project can meet all formal QA requirements and still be of low quality (or flatly erroneous). On the other hand high quality work can be done without a formal QA system.

The guidelines given below provide examples of the sorts of issues that should be dealt with in a formal QA management system. The issues covered are based on the process of carrying out a CFD simulation as outlined in Chapter 2.

4.6.2.1. Guidelines on problem definition

- The user needs to give careful thought to the requirements and objectives of the simulation and typically might consider the following points:
 - Is a CFD simulation method really appropriate (e.g. for wave driven problems, is the RANSE approach most appropriate?)?
 - Are the objectives of the simulation clearly defined?
 - What are the requirements on accuracy?
 - What local/global quantities are needed from the simulation?
 - What are the documentation/reporting requirements?
 - What are the important flow physics involved (steady, unsteady, single phase, laminar, turbulent, transitional, internal, external, etc.)?
 - What is the area of primary interest (domain) for the flow calculation?
 - Is the geometry well defined?
 - What level of validation is necessary? Is this a routine application, where validation and calibration has already been carried out on similar flow fields, and where only relatively small changes can be expected from earlier similar simulations? Or is it a non-routine application, where little earlier validation work has been done.
 - What level of computational resources is needed for the simulation (memory, disk space, CPU time) and are these available?

4.6.2.2. Guidelines on solution strategy

- Having established a clear problem definition, the user needs to translate this into a solution strategy involving issues and questions that have been addressed in the earlier chapters of this document, such as:
 - Mathematical and physical models.
 - Pressure or density based solution method.

- Turbulence model.
- Available code/solver.
- Computational mesh.
- Boundary conditions.

4.6.2.3. Guidelines on code-handling

- A potential source of user errors is in implementing the solution strategy with a particular code. Such errors might be minimised by the availability of a formal check list or by letting another CFD analyst checking through the code input data. The types of questions which should be considered are:
 - Have the boundary conditions not only been properly defined, but also properly applied?
 - Has the appropriate system of units been used?
 - Is the geometry correct?
 - Are the correct physical properties specified?
 - Have the intended physical and mathematical models been used (e.g. gravity forces, rotation, user defined functions)?
 - Have default parameters been changed which may affect the solution?
 - Has the appropriate convergence criterion been defined and used?

4.6.2.4. Guidelines on interpretation

- Don't be seduced into believing that the solution is correct just because it has converged and produced high-quality colour plots (or even seductive video presentations) of the CFD simulations. Make sure that an elementary interpretation of the flow-field explains the fluid behaviour and that the trends of the flow analysis can be reconciled with a simple view of the flow.
- Make sure that the mean values of engineering parameters derived from the simulation are computed consistently (e.g. mass-average values, area-average values, time-average values). Calculation of local and mean engineering parameters with external post-processing software may be inconsistent with the solution method of the code used (e.g. calculating shear stresses from the velocities, calculating shear stresses using nodal values instead of wall functions). Check that any test data used for comparison with the simulations is also computed in the same way as the data from the simulation.
- Consider whether the interpretation of the results and any decisions made, is within the accuracy of your computation.

4.6.2.5. Guidelines on documentation

- Keep good records of the simulation with clear documentation of assumptions, approximations, simplifications, geometry and data sources.
- Organise the documentation of the calculations so that another CFD expert can follow what has been done.
- Be aware that the level of documentation required depends strongly on the customers requirements as defined in the problem definition.

4.6.3. Training requirements for CFD users

The growth in the use of CFD codes and the trend for them to become rich packages with lots of alternative modelling options, steadily increases the risk of user errors. This trend is reinforced by the ease of use of modern computer codes with simple graphical user interfaces making them available for inexperienced users. Although efforts are taken to simplify the usage of CFD codes, careful training with realistic exercises should still be considered as the starting point of any user's CFD career. The theoretical part of the training should focus on fundamental modelling features, their underlying assumptions and their limitations. The same information is also a central part of a good user documentation. Unlike linear finite element

stress analysis, CFD still requires expertly trained users for good results. In situations where non-experienced users have to be used, some restriction on their freedom to adjust critical parameters might be advisable, and they should be limited to simulations of routine types.

Depending on the CFD software, additional training on grid generation is advisable.

4.6.3.1. Guidelines

- A CFD user for non-routine applications should have good training and knowledge in classical fluid mechanics, a broad understanding of numerical methods, and detailed knowledge of the application being examined. This means that they will be able to understand the limitations of the models used (e.g. turbulence, radiation, buoyancy driven flows).
- The training and education requirement for more routine applications can be less stringent, provided that clear guidelines or procedures have been established for the use of the code being used. An example of a routine application would be the simulation of a standard component in a design environment where many previous designs have been calculated and only relatively small changes in geometries and boundaries conditions occur.
- In both routine and non-routine applications, training on the use of the specific CFD code with the solution of realistic exercises is needed.

4.7. Code errors

The success of a code generally leads to it becoming used by more users. As the user-base expands, there are increasing demands for more options and the code becomes more and more complex. As it can deal with more difficult problems, there is again an expansion of its use. In the end it is inevitable that code errors will be discovered by many users who outnumber the developers by an order of magnitude and have a much wider range of applications and test cases than the code developers themselves.

The size and complexity of large CFD software packages inevitably mean that code errors (bugs) may still be present in the software even if it has been in use and development for many years. The painstaking but straightforward process of verification provides a means of checking that the code faithfully reproduces the model approximations incorporated in the algorithms being programmed. The main problem associated with code verification is that the accuracy of a code can never be formally demonstrated for all possible conditions and applications, and for all possible combinations of valid code input options. In fact it can never be proven that a code is correct in that sense, as at any time a new bug may be found.

4.7.1.1. Guidelines for the code developer and vendor

- The code developer or code vendor needs to demonstrate that he has applied stringent methods of quality control to the software development and maintenance.
- Verification of the code is to be carried out by the code vendor or developer, and he should provide the necessary information on the verification process for the user.
- The code developer or code vendor should maintain and publish a databank of verification test cases that are used for testing. The cases should include simple code verifications tests (e.g. that solutions are independent of co-ordinate systems).
- The code developer or code vendor should provide documentary evidence of the verification tests that the software has undergone, which should include clearly details of the code options which are used during testing.
- For all new versions of the code a standard set of verification test cases should be repeated.
- Code vendors and code developers should supply a list of known bugs and errors in each version of the code (hot-line, password secured web-page). This list should demonstrate that the number of bugs reduces as the code matures.

- The code developers should try to include warning notices and guidance for the user in the output. For example, when basic rules on grid generation (expansion ratios, skew, etc.) are being broken, when important specific default options are being overruled by the code input data or when the near wall grid is inconsistent with the turbulence modelling.

4.7.1.2. Guidelines for the code user

- The user should recognise that codes can only be validated and verified for a class of problems involving specific variables. If the user is moving into an area where the code is not fully verified there is more risk of code errors.
- A suite of test cases set up and run by the user on new code releases provides an independent check on the code and highlights changes between releases (for example in default parameters).
- When a code error is suspected, the user should communicate this to the code vendor or developer as soon as possible, especially if no list of known bugs has been published. Other users may then profit from this experience or the user may find that the bug is well-known and a solution or work-around is available.
- In communication with the code developer or code vendor about a suspected program error, the user should provide a short concise description of the problem and all the necessary input data files so that the error can be reproduced. In cases where commercial sensitivity precludes this, special arrangements will need to be made.

5. Potential flow and diffraction calculations

5.1. General guidance in panel mesh generation

The body surface is usually modelled using combinations of quadrilateral and triangular panels. It is important to remember that the panels are not physical but actually represent distributions of sources, vortices or dipoles. Distributions can take the following forms:

- A single point source distribution is used for simplicity where the distance to the panel source is large, as a refined distribution will not produce a more accurate result.
- A cluster of point sources improves the accuracy over a single point source and is still simpler and faster than a true surface distribution.
- A plane panel of constant source strength was suggested by Hess and Smith [1964] and is used by many present day codes.
- A non-planar panel of constant source strength, made up of triangular elements, was developed by Jensen [1988].
- A curved panel with a bilinearly varying source strength was developed by Wei (1987) and is used in some higher order methods.
- Methods based on spline representations of both the surface and the potential are also now in use (Newman)

For most ship flows, flat panels give sufficient accuracy and are simpler to construct.

The source strength must be continuous across the panel joints, except at a trailing edge. Here the Kutta condition must be satisfied, such that the flow can't go around the trailing edge, but must leave the body there.

Many basic panel codes use flat, planar panels with the vertices located on the body. A common method of curved surface panel generation uses a parametric cubic spline or NURBS procedure to approximate the true body curve. Automatic panel generation programs are widely available for this. The facility to define a number of bodies or separate parts of the same body independently allows complex bodies and flows to be investigated.

For steady free surface applications the number of panels required for accurate calculation of the wave profile is inversely proportional to the square of the Froude number. The lower the speed range to be investigated, the more panels there will be in the longitudinal free surface mesh.

Guidelines:

- Ensure that panels edges meet exactly and that the body is totally enclosed, especially if importing body geometry from a CAD model.
- Grid refinement is required in areas of rapid pressure change.
- Flow separation will only occur wherever the user sets it to (i.e. where a wake sheet is applied).
- Careful panel definition is required at regions of high curvature (e.g. at the leading edge of propeller blades, fin stabilisers) to represent the body accurately. A finer distribution of panels should be used in regions likely to experience high fluid flow.
- The trailing edge must be located at a panel intersection to satisfy the Kutta condition. When defining panels around a section it may be easiest to start from the trailing edge.
- If the panels or the fluid domain are to be translated or rotated careful thought should be given to the location of the panels.
- If a cubic spline formulation is used care needs must be taken with the curve end conditions when trying to model sharp changes in direction.
- Adjacent bodies must not intersect or overlap.

- Panels should have a low aspect ratio and should not be highly skewed. Element sizes should vary gradually over the body. Should quadrilateral panels exhibit high levels of skew, they should be replaced by two triangular panels, blended to the surrounding panel size.
- Plate element normals must point outwards from the body.
- Try to use the symmetry properties of the body geometry to the full.
- For free surface flows at least 30 panels per wavelength are required for adequate resolution of the wave profile, and users should in any case perform mesh sensitivity studies to gain confidence in the results.
- The wake sheet should extend far enough downstream to capture sufficient detail of the flow.
- For propellers, the optimum chord-wise panel distribution will depend on the shape and radius of the leading edge.

5.2. Definition of boundary conditions

The inherent assumptions of potential flow methods are that the flow is:

- inviscid;
- irrotational;

The conditions imposed on the disturbance potential ϕ are that:

- the velocity potential satisfies Laplace's equation everywhere outside of the body and the wake;
- the disturbance potential due to the body vanishes at infinity;
- the normal component of velocity is zero on the body surface;
- the Kutta condition of a finite velocity at the trailing edge is satisfied;
- the wake sheet is a stream surface with equal pressure either side.

For free surface flows the following boundary conditions are also invoked:

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

Equilibrium condition: The weight and external forces acting on the body are in equilibrium.

Dynamic boundary condition: Atmospheric pressure acts at the water surface.

Radiation boundary condition: This depends upon the forward speed of the vessel, depth of water, and the problem type. For the infinite water depth, steady forward speed problem, waves exist only in the sector behind the ship and do not propagate ahead. As the water depth decreases, and in very shallow water, issues relating to depth effects may become important. For sea-keeping problems at forward speed, care needs to be taken with the combination of ship speed and wave frequencies used (the important parameter is $U\omega/g$).

Domain boundary conditions: Waves generated by the vessel should pass out of the computational domain without reflection.

Guidelines:

- Check that appropriate boundary conditions are available for the flow being modelled
- Ensure that waves are not reflected from the domain boundaries.
- Systematic variation of boundary conditions e.g. the location of a radiation boundary, should be carried out to determine the uncertainty effects. If these effects are significant a more detailed analysis of the boundary conditions will be necessary.

- The wall boundary conditions will inherently be “free-slip” for a potential flow. If this is unsuitable, a different method or different viscous approximation should be used.

5.3. Special considerations for non-linear methods

Many of the phenomena associated with a ship or free structure in a seaway will be non-linear (e.g. roll damping, motions of water on deck, added resistance in waves). Most of these effects cannot be accounted for by even second or third order linear perturbation methods. The traditional method of dealing with these non-linear effects is by linearising the motion and solving the motion equations in the time domain. Time integration is performed by standard methods such as Euler, Runge-Kutta or predictor-corrector methods.

Guidelines:

- Linearised potential flow methods have limitations with regard to wave slope.
- Careful panel distribution is required at the vessel/free surface interface to provide enough resolution to resolve the wave profile sufficiently.
- Should wave breaking be possible within the solution, for example near the bow or at high speed, solutions may be unstable and require local grid coarsening to achieve a converged result.
- Control of the free surface panel size in the far field should take account of the effect of growing panel size on wave propagation and speed.

5.4. Integration of viscous effects

5.4.1. General

Potential flow methods offer a convenient and relatively simple way of determining general flow patterns and forces around arbitrary bodies. The main drawback is the inherent neglect of viscous effects. In many applications where a potential flow method is used, such as in the high Reynolds number regime for ship hulls, the viscous effects will be confined to thin attached boundary layers, wakes and regions of free shear. For problems such as the solution of roll damping, viscous effects will become large. There are a number of ways of including some estimation of the viscous effects.

5.4.2. Empirical and semi-empirical methods

A number of empirical techniques have been developed to include the effects of the boundary layer with regard to skin friction and the alteration to the pressure distribution around the body. A fully empirical approximation to the skin friction drag can be used (e.g. the ITTC 1957 correlation line). In this case the local skin friction for each panel can be calculated based on a parametric length from the leading edge or stagnation point. The form drag can be approximated by assuming that the important region for viscous shear is confined to a narrow domain next to the body and the trailing wake. The displacement thickness of the boundary layer is calculated and the corresponding panels then displaced by this amount.

One problem with such formulations is that it is hard to prescribe the correct form of viscous correction equations for effects such as stern wave systems, as this is highly sensitive to hull form shape. A second problem is that traditional ship correlation lines already contain some effects associated with hull form. These techniques are no longer in frequent use.

5.4.3. Solution of Navier Stokes equations

An alternative to empirical techniques is to solve the RANS equations in the domain surrounding the vessel and with the potential flow solution used to define the shape and location of the free surface. This is currently only considered suitable for steady ship flow problems. The free surface boundary is treated as a free-slip wall, and viscous effects at the

free surface assumed to be negligible. The main benefit of the method is that it allows the full wetted surface of the vessel to be included in the calculations.

Guidelines:

- The use of empirical formula to estimate additional viscous effects should be used as an approximate method only, and care should be exercised in the choice of skin friction correlation line.
- Such methods can only be applied where the flow remains attached.
- For accurate resolution of stern wave and transom effects, where viscous forces are significant, empirical viscous approximations may not be sufficient.

6. Viscous turbulent flows

6.1. Solution algorithm

The discretised set of RANS equations can be solved with various solution procedures such as either pressure-based and or density-based methods (for a review, see Ferziger and Peric [1998], Fletcher [1991] or Hirsch [1991]). The solution algorithms make use of numerous tuning parameters, such as artificial time-steps, under-relaxation, etc., to improve convergence behaviour and robustness of the code. The field of application of a code and the modelling technique included influence the choice of the numerical method and the solution procedure. In principle the solution of a well converged simulation is independent of the numerical method and the solution algorithm chosen.

Guidelines

- Check the adequacy of the solution procedure with respect to the physical properties of the flow.
- As a first step in this process, the parameters controlling convergence (e.g. relaxation parameters or Courant number) of the solution algorithm should be used as suggested by the CFD-code vendor or developer.
- If it is necessary to change parameters to aid convergence, it is not advisable to change too many parameters in one step, as it then becomes difficult to analyse which of the changes have influenced the convergence. In case of persistent divergence see sections on boundary conditions (section 3.7), grid (section 3.4), discretisation and convergence errors (section 3.2).
- Consider carefully whether the flow can be expected to exhibit a steady or unsteady flow behaviour. Consider the size of the unsteady scales to be expected present in the flow field in comparison to the geometrical dimensions, and if this is large then an unsteady simulation is necessary.
- If a steady solution has been computed and there is a reason to be unsure that the flow is really steady, then an unsteady simulation should be carried out with the existing steady flow field as the initial condition. Examination of the time-development of the physical quantities in the locations of interest will identify whether the flow is steady or not.

6.2. Turbulence modelling

Most flows of practical engineering interest are turbulent, and the turbulent mixing of the flow then usually dominates the behaviour of the fluid. The turbulent nature of the flow plays a crucial part in the determination of many relevant engineering parameters, such as frictional drag, flow separation, transition from laminar to turbulent flow, thickness of boundary layers, extent of secondary flows, and spreading of jets and wakes.

The turbulent states which can be encountered across the whole range of industrially relevant flows are rich, complex and varied. After a century of intensive theoretical and experimental research, it is now accepted that no single turbulence model can span these states and that there is no generally valid universal model of turbulence. A bewildering number and variety of models have appeared over the years, as different developers have tried to introduce improvements to the models that are available. The extremely difficult nature of this endeavour caused Bradshaw [1994] to refer to turbulence as "the invention of the Devil on the 7th day of creation, when the Good Lord wasn't looking".

The available turbulence models can be roughly divided into four main categories:

- Algebraic (or zero-equation) models
- One-equation models
- Two-equation models

- Stress transport models

and within each of these categories there are a wide variety of different models and options available (see below). The choice of which turbulence model to use and the interpretation of its performance (i.e. establishing bounds on key predicted parameters) is a far from trivial matter.

A set of application procedures is evidently required which documents the performance of various turbulence models across a broad class of flow regimes and for different applications. A full categorisation of this type is beyond the scope of the present guidelines. Instead the general features and broad limitations of different classes of model will be discussed and guidance will be given on the practical deployment of the turbulence model most commonly used in industrial practice, the standard k- ϵ model. A fuller introduction to the subject can be obtained by consulting standard reference texts on the subject, such as Launder and Spalding [1972], Cebeci and Smith [1974], Rodi [1981], Patankar [1980], Tennekes and Lumley [1972] and Wilcox [1998].

6.2.1. RANS equations and turbulence models

Turbulent flows contain many unsteady eddies covering a range of sizes and time scales. For flows in industrial applications, the effects of turbulence are examined using the so-called (Reynolds-averaged Navier-Stokes) RANS equations. These are developed from the time-dependent three-dimensional Navier-Stokes equations which are averaged in such a manner that unsteady structures of small sizes in space and time are eliminated and become expressed by their mean effects on the flow through the so-called Reynolds or turbulent stresses. These stresses must be interpreted in terms of calculated time-averaged variables in order to close the system of equations thereby rendering them solvable. This requires the construction of a mathematical model known as a turbulence model, involving additional correlations for the unknown quantities.

Because models are based on different assumptions, all available turbulence models have limitations which depend on the modelling strategy.

6.2.2. Classes of turbulence models

6.2.2.1. Eddy viscosity models

The simplest turbulence modelling approach rests on the concept of a turbulent viscosity, μ_T . This relates the turbulent stresses appearing in the RANS equations to the gradients of time-averaged velocity (i.e. the rate of strain) in direct analogy to the classical interpretation of viscous stresses in laminar flow by means of the fluid viscosity, μ . Thus for example, in a shear layer where the dominant velocity gradient is $\partial u/\partial y$ (u is time-averaged velocity in the principal direction of flow and y is the cross-stream co-ordinate) the turbulent shear stress is given as $\rho \cdot \mu_T \cdot \partial u/\partial y$.

From dimensional considerations, μ_T/ρ is proportional to $V \cdot L$, where V is a velocity scale and L is a length scale of the larger turbulent motions (often called the mixing length). Both the velocity scale V and the length scale L are determined by the state of turbulence, and, over the years, various prescriptions for V and L have been proposed.

6.2.2.2. Algebraic (or zero-equation) models

The simplest prescription of V and L is with the so called algebraic (or zero-equation) class of models. These assume that V and L can be related by algebraic equations to the local properties of the flow, see, for example, Cebeci and Smith [1974] and Baldwin and Lomax [1978]. For example, in a wake or free shear layer V is often taken as proportional to the velocity difference across the flow and L is taken as constant and proportional to the width of the layer. In a boundary layer close to the wall V is given as $L \cdot \partial u/\partial y$ (or $L \cdot \Omega$ where Ω is the magnitude of the vorticity) and L is related to the wall-normal distance from the wall (y -direction). The outer part of the boundary layer is treated in a similar manner to a wake. The

turbulent Prandtl number is given a constant value close to unity except very close to a wall where viscous effects become important.

Algebraic models of turbulence have the virtue of simplicity and are widely used with considerable success for simple shear flows such as attached boundary layers, jets and wakes. For more complex flows where the state of turbulence is not locally determined but related to the upstream history of the flow, a more sophisticated prescription is required.

6.2.2.3. One-equation models

The one-equation models attempt to improve on the zero-equation models by using an eddy viscosity that no longer depends purely on the local flow conditions but takes into account where the flow has come from, i.e. upon the flow history. The majority of approaches seek to determine V and L separately and then construct μ_T/ρ as the product of V and L . Almost without exception, V is identified with $k^{1/2}$, where k is the kinetic energy per unit mass of fluid arising from the turbulent fluctuations in velocity around the time-averaged velocity. A transport equation for k can be derived from the Navier Stokes equations and this is the single transport equation in the one-equation model. This is closed (i.e. reduced to a form involving only calculated variables) by introducing simple modelling assumptions thereby furnishing a robust prescription for V which accounts for non-local effects. It is then possible to algebraically prescribe L with reasonable confidence (e.g. in regions close to a wall) whilst solving the k -equation for the velocity scale V .

Spalart and Allmaras [1992] have devised an alternative formulation of a one-equation model which determines the turbulent viscosity directly from a single transport equation for μ_T and this model is proving very successful for practical turbulent flows in aerospace applications, particularly in the USA.

6.2.2.4. Two-equation models

For general applications, it is usual to solve two separate transport equations to determine V and L , giving rise to the name two-equation model. In combination with the transport equation for k , an additional transport equation is solved for a quantity which determines the length scale L . This class of models (two-equation models) is the most commonly used in industrial application since it is the simplest level of closure which does not require geometry or flow regime dependent input.

The most popular version of two equation models is the k - ϵ model, where ϵ is the rate at which turbulent energy is dissipated by the action of viscosity on the smallest eddies (Launder and Spalding [1974]). A modelled transport equation for ϵ is solved and then L is determined as $C_\mu k^{3/2}/\epsilon$ where C_μ is a constant. The second most widely used type of two equation model is the k - ω model, where ω is a frequency of the large eddies (Wilcox [1998]). A modelled transport equation for ω is solved and L is then determined as $k^{1/2}/\omega$.

The k - ω model performs very well close to walls in boundary layer flows, particularly under strong adverse pressure gradients. However it is very sensitive to the free stream value of ω and unless great care is taken in setting this value, spurious results are obtained in both boundary layer flows and free shear flows. The k - ϵ model is less sensitive to free stream values but generally inadequate in adverse pressure gradients and so Menter [1993, 1994a, 1994b, 1996] has proposed a model which retains the properties of k - ω close to the wall and gradually blends into the k - ϵ model away from the wall. This model has been shown to eliminate the free stream sensitivity problem without sacrificing the k - ω near wall performance.

The performance of two-equation turbulence models deteriorates when the turbulence structure is no longer close to local equilibrium. This occurs when the ratio of the production of turbulence energy to the rate at which it is dissipated at the small scales (i.e. ϵ) departs significantly from its 'equilibrium value', or equivalently when dimensionless strain rates (i.e. absolute value of the rate of strain times k/ϵ) become large. Various attempts have been made to modify two equation turbulence models to account for strong non-equilibrium effects.

For example, the so-called SST (shear stress transport) variation of Menter's model, Menter [1993, 1996], leads to marked improvements in performance for non-equilibrium boundary layer regions such as may be found close to separation.

6.2.2.5. Reynolds stress transport models

The two-equation turbulence models described above presume that the turbulent stresses are linearly related to the rate of strain by a scalar turbulent viscosity, and that the principal strain directions are aligned to the principal stress directions. This is reasonable for fairly simple states of strain, especially when the model constants have been carefully calibrated from similar classes of flows, but may prove totally inadequate for modelling complex strain fields arising from the action of swirl, body forces such as buoyancy or extreme geometrical complexity. Under such circumstances a more subtle relationship between stress and strain must be invoked. The so called Reynolds stress transport models (RSM) dispense with notion of turbulent viscosity, and determine the turbulent stresses directly by solving a transport equation for each stress component, requiring the solution of six additional coupled equations, together with an equation for ϵ to provide a length scale (Launder and Spalding [1972], Rodi [1981], Launder et al. [1975] and Speziale [1987a]). In a similar way, the turbulent heat fluxes can be determined directly by solving three extra equations, one for each flux component thereby removing the notion of turbulent Prandtl number.

This form of model can handle complex strain and, in principle, can cope with non-equilibrium flows. However, it is complex, expensive to compute, can lead to problems of convergence and also requires boundary conditions for each of the new parameters being solved. For these reasons it has not yet been widely adopted as an industrial tool.

6.2.2.6. Other models

An alternative, somewhat simpler approach for dealing with complex strain is provided by the non-linear eddy viscosity class of models, see for example, Apsley et al. [1997]. These models retain the idea that the turbulent stresses can be algebraically related to the rate of strain (i.e. time averaged velocity gradients), but higher order quadratic and cubic terms are included. Such models are gaining in popularity since they involve the same number of equations as two equation models and thus are computationally efficient.

Some turbulence models are valid for the turbulent flow region, but fail in the laminar viscous sub-layer close to the wall. Various so-called low-Reynolds number versions of the k- ϵ and RSM models have been proposed incorporating modifications which remove this limitation (Patel et al. [1985] and Wilcox [1998]). Alternatively the standard k- ϵ and RSM models can be used in the interior of the flow and coupled to a one-equation model which is used to resolve just the wall region. This is known as a two-layer model.

Many other models have been developed which are not referred to here, and for further details the interested reader is referred to the very extensive literature on this subject.

6.2.2.7. Guidelines on turbulence modelling

- The user should be aware that there is no universally valid general model of turbulence that is accurate for all classes of flows. Validation and calibration of the turbulence model is necessary for all applications.
- If possible, the user should examine the effect and sensitivity of results to the turbulence model by changing the turbulence model being used.
- The relevance of turbulence modelling only becomes significant in CFD simulations when other sources of error, in particular the numerical and convergence errors, have been removed or properly controlled. Clearly no proper evaluation of the merits of different turbulence models can be made unless the discretisation error of the numerical algorithm is known, and grid sensitivity studies become crucial for all turbulence model computations.

6.3. Weaknesses of the standard $k-\epsilon$ model

Despite the great variety of turbulence modelling options available to the user, the standard $k-\epsilon$ model with wall functions, as set out by Launder and Spalding [1974] remains the work-horse of industrial computation. It is therefore of value to catalogue the major weaknesses associated with this model in practical application and, where possible, indicate palliative actions which might be fruitfully considered. These are listed below. The advisory actions are drawn from an extensive literature on the subject and should not to be viewed as definitive cures. The manuals of commercial and in-house codes may proffer alternative and equally effective advice, and many commercial codes will include alternatives to the standard $k-\epsilon$ model. Where the action given below involves a modification or adjustment to the standard $k-\epsilon$ model, this should be regarded as specific palliative for the weakness under consideration and will not usually prove of general benefit (and may even be worse).

6.3.1. Guidelines on weaknesses of the standard $k-\epsilon$ model

- The turbulent kinetic energy is over-predicted in regions of flow impingement and re-attachment leading to poor prediction of the development of flow around leading edges and bluff bodies. Kato and Launder [1993] have proposed a modification to the transport equation for ϵ which is designed to tackle this problem.
- Regions of re-circulation in a swirling flow are under-estimated. Reynolds Stress models (RSM) should be used to overcome this problem.
- Highly swirling flows are generally poorly predicted due to the complex strain fields. Reynolds Stress models (RSM) or non-linear eddy viscosity models should be used in these cases.
- Mixing is poorly predicted in flows with strong buoyancy effects or high streamline curvature. Reynolds Stress models should be used in these cases.
- Flow separation from surfaces under the action of adverse pressure gradients is poorly predicted. The real flow is likely to be much closer to separation (or more separated) than the calculations suggest. The Baldwin-Lomax one-equation model is often better than the standard $k-\epsilon$ model in this respect, Baldwin and Lomax [1978]. The SST version of Menter's $k-\omega$ based, near wall resolved model mentioned in section 4.2.4 (Menter [1993, 1996]) also offers a considerable improvement.
- Flow recovery following re-attachment is poorly predicted. Avoid the use of wall functions in these regions.
- The spreading rates of wakes and round jets are predicted incorrectly. The use of non-linear $k-\epsilon$ models should be investigated for these problems.
- Turbulence driven secondary flows in straight ducts of non-circular cross section are not predicted at all. Linear eddy viscosity models cannot capture this feature. Use RSM or non-linear eddy viscosity modelling.
- Laminar and transitional regions of flow cannot be modelled with the standard $k-\epsilon$ model. This is an active area of research in turbulence modelling. No simple practical advice can be given other than advocating user intervention to switch the turbulence model on or off at predetermined locations.

6.4. Near wall modelling

In wall attached boundary layers, the normal gradients in the flow variables become extremely large as wall distance reduces to zero. A large number of mesh points packed close to the wall is required to resolve these gradients. Furthermore, as the wall is approached, turbulent fluctuations are suppressed and eventually viscous effects become important in the region known as the viscous sub-layer. This modified turbulence structure means that many standard turbulence models (see summary given above) are not valid all the way through to the wall. Thus special wall modelling procedures are required.

6.4.1. Wall functions

This is the procedure most commonly used in industrial practice. The difficult near-wall region is not explicitly resolved with the numerical model but is bridged using so called wall functions (Rodi [1981] and Wilcox [1998]). In order to construct these functions the region close to the wall is characterised in terms of variables rendered dimensionless with respect to conditions at the wall.

The wall friction velocity u_τ is defined as $(\tau_w/\rho)^{1/2}$ where τ_w is the wall shear stress. Let y be normal distance from the wall and let U be time-averaged velocity parallel to the wall. Then the dimensionless velocity, U^+ and dimensionless wall distance, y^+ are defined as U/u_τ and $y \cdot \rho \cdot u_\tau / \mu$ respectively. If the flow close to the wall is determined by conditions at the wall then U^+ can be expected to be a universal function of y^+ up to some limiting value of y^+ . This is indeed observed in practice, with a linear relationship between U^+ and y^+ in the viscous sub-layer, and a logarithmic relationship, known as the law of the wall, in the layers adjacent to this (so-called log-layer). The y^+ -limit of validity depends on external factors such as pressure gradient and the penetration of far field influences. In some circumstances the range of validity may also be effected by local influences such as buoyancy forces if there is strong heat transfer at the wall. The turbulence velocity ($k^{1/2}$) and length scales, when treated in the same way also exhibit a universal behaviour.

These universal functions can be used to relate flow variables at the first computational mesh point, displaced some distance y from the wall, directly to the wall shear stress without resolving the structure in between. The only constraint on the value of y is that y^+ at the mesh point remains within the limit of validity of the wall functions. A similar universal, non-dimensional function can be constructed which relates the temperature difference between the wall and the mesh point to heat flux at the wall (Rodi [1981]). This can be used to bridge the near-wall region when solving the energy equation.

6.4.2. Wall function guidelines

- The meshing should be arranged so that the values of y^+ at all the wall adjacent mesh points is greater than 30 (the form usually assumed for the wall functions is not valid much below this value). It is advisable that the y^+ values do not exceed 100 and should certainly never be less than 11. Some commercial CFD codes account for this by switching to alternative functions if y^+ is < 30 . Be aware of this and check the user manuals.
- Cell centred schemes have their integration points at different locations in a mesh cell than cell vertex schemes. Thus the y^+ value associated with a wall adjacent cell differs according to which scheme is being used on the mesh. Care should be exercised when calculating the flow using different schemes or codes with wall functions on the same mesh.
- The values of y^+ at the wall adjacent cells strongly influence the prediction of friction and hence drag. Thus particular care should be given to the placement of near-wall meshing if these are important elements of the solution.
- Check that the correct form of the wall function is being used to take into account the wall roughness.

6.4.3. Near wall resolution

As already mentioned, a universal near-wall behaviour over a practical range of y^+ may not be realisable everywhere in a flow. Under such circumstances the wall-function concept breaks down and its use will lead to significant error, particularly if wall friction and heat transfer rates are important. The alternative is to fully resolve the flow structure through to the wall. Some turbulence models can be validly used for this purpose, others cannot. For example, the $k-\omega$ two-equation model can be deployed through to the wall as can the one-equation $k-L$ model (e.g. Wolfstein [1969]). The standard $k-\epsilon$ and RSM models cannot. Various so-called low-Reynolds number versions of the $k-\epsilon$ and RSM models have been proposed incorporating modifications which remove this limitation (Patel et al. [1985] and Wilcox [1998]). Alternatively

the standard k- ϵ and RSM models can be used in the interior of the flow and coupled to the k-L model which is used to resolve just the wall region. This is known as a two-layer model. Whatever modelling approach is adopted, a large number of mesh points must be packed into a very narrow region adjacent to the wall in order to capture the variation in the flow variables.

6.4.4. Near wall resolution guidelines

- Make sure that the turbulence model being used is capable of resolving the flow structure through to the wall.
- The value of y^+ at the first node adjacent to the wall should be close to unity.
- Employ a small stretching factor for progressing the mesh spacing away from the wall. There should be at least ten mesh points between the wall and y^+ equal to 20.

6.5. Inflow boundary conditions

The use of a turbulence model (other than an algebraic model) requires that turbulence properties at a domain inlet region need to be specified. Verified quantities should be used as inlet boundary conditions for k and ϵ , because the magnitude can significantly influence the results. If there are no data available, then the influence of the choice should be examined by sensitivity tests with different simulations.

6.6. Unsteady flows

The use of the RANS equations in an unsteady flow is valid, provided that the large scale eddies are smaller than the order of the geometry itself. If the unsteadiness is provoked from an external source (such as due to wakes or wave motions) the turbulence model does not then interact with the frequency and amplitude of the unsteadiness, and provided that the time-scale of the unsteadiness is sufficiently far removed from the turbulence scales, then the use of turbulence modelling is acceptable. If the unsteadiness is self-induced, such as vortex shedding from a bluff body, then difficulties with turbulence modelling may occur. See also section 3.6 on temporal discretisation errors.

6.7. Laminar and transition flows

The distinction between laminar, transitional and turbulent flow is difficult. Sometimes the flow appears in different states depending on the location of the area of interest, for example, the flow in an inlet of a machine can be laminar whereas the flow inside the machine is turbulent. Another example is the flow over an airfoil, which is normally laminar at the leading edge and turbulent in the wake behind. The general problem of the transition from laminar to turbulent flow and the computation of the origin of turbulence is a subject of fundamental academic research. It cannot be included in general industrial CFD-computations.

The simplest way around this problem is to calculate the flow as a turbulent one. The turbulent kinetic energy is approximately zero in the nominal laminar flow regimes. Special care needs to be taken if a turbulence model with wall function is being used to get information about wall shear stress.

6.7.1. Guidelines

- Check that the flow does not contain extensive regions of laminar or transition flow that would be incorrectly estimated by the k- ϵ turbulence model.

6.8. Mesh generation

The computational grid represents the geometry of the region of interest. It consists of grid cells that provide an adequate resolution of the geometrical features. In hydrodynamics, body-fitted grids are used almost universally. However, several kinds of mesh topology are available:

- Structured grid: The points of a block are addressed by a triple of indices (ijk). The connectivity is straight-forward because cells adjacent to a given face are identified by the indices. Cell edges form continuous mesh lines which start and end on opposite block faces. Cells have the shape of hexahedral, but a small number of prisms, pyramids and tetrahedra with degenerated faces and edges are sometimes accepted.
- Block structured grid: For the sake of flexibility the mesh is assembled from a number of structured blocks attached to each other. Attachments may be regular, i.e. cell faces of adjacent blocks match, or arbitrary (general attachment without matching cell faces).
- Chimera grid: Structured mesh blocks are placed freely in the domain to fit the geometrical boundaries and to satisfy resolution requirements. Blocks may overlap, and instead of attachments at block boundaries information between different blocks is transferred in the overlapping region.
- Unstructured grid: Meshes are allowed to be assembled cell by cell freely without considering continuity of mesh lines. Hence, the connectivity information for each cell face needs to be stored in a table. The most typical cell shape is the tetrahedron, but any other form including hexahedral cells is possible.
- Hybrid grid: This grid combines structured with unstructured meshes.

The grid must be fine enough to capture all important flow features. This may be achieved by local grid refinement. Unstructured meshes are especially well suited for this purpose. If block structured grids are used local refinement results in block attachments with dissimilar number of grid lines. Some CFD codes provide algorithms to adapt the grid resolution locally according to numerical criteria from the flow solution, such as gradient information or error estimators.

The accuracy of the simulation increases with increasing number of cells, i.e. with decreasing cell size. However, due to limitations imposed by the increased computer storage and run-time some compromise is nearly always inevitable.

In addition to grid density, the quality of a mesh depends on various criteria such as the shape of the cells (aspect ratio, skewness, included angle of adjacent faces), distances of cell faces from boundaries or spatial distribution of cell sizes. The introduction of special topological features such as O-grids or C-grids and care taken to locate block-interfaces in a sensible manner can help to improve the overall quality of a block-structured mesh. Unstructured meshing techniques may take advantage of prism layers with structured submeshes close to domain boundaries.

Guidelines

- Clean up CAD geometry and for body fitted grids check that the surface grid conforms to the CAD geometry (see also Section 6.1.1).
- When using periodic boundary conditions ensure high precision of the interface.
- Avoid highly skewed cells, in particular for hexahedral cells or prisms the included angles between the grid lines should be optimised in such a way that the angles are approximately 90 degrees. Angles with less than 40 or more than 140 degrees often show a deterioration in the results or lead to numerical instabilities, especially in the case of transient simulations.
- The angle between the grid lines and the boundary of the computational domain (the wall or the inlet- and outlet-boundaries) should be close to 90 degrees. This requirement is stronger than the requirement for the angles in the flow field far away from the domain boundaries.
- Avoid the use of tetrahedral elements in boundary layers.
- Away from boundaries, ensure that the aspect ratio (the ratio of the sides of the elements) is not too large. This aspect ratio should be typically not larger than 20. Near walls this restriction may be relaxed and indeed can be beneficial.

- The code requirements of mesh stretching or expansion ratios (rates of change of cell size for adjacent cells) should be observed. The change in mesh spacing should be continuous and mesh size discontinuities be avoided, particularly in regions of high gradients.
- The mesh should be finer in critical regions with high flow gradients, such as regions with high shear, and where there are significant changes in geometry or where suggested by error estimators. Make use of local refinement of the mesh in these regions, in accordance with the selected turbulence wall modelling (see Section 5.3). The location of a refinement interface should be away from high flow gradients.
- Check the assumption of regions of high flow gradients assumed for the grid with the result of the computation and rearrange grid points if found to be necessary.
- Analyse the suitability of the mesh by a grid dependency study (this could be local) where you use at least three different grid resolutions. If this is not feasible try to compare different order of spatial discretisations on the same mesh (see Section 3.5). The ITTC guidelines provide more detail in this area.
- Use the global topology of the mesh to help satisfy the above guidelines.

6.9. Choice of boundary conditions

Two types of boundary conditions and combinations of them are most commonly encountered. The Dirichlet condition specifies the distribution of a physical quantity over the boundary at a given time step and the Neumann condition defines the distribution of its first derivative.

Users have normally no control on the spatial discretisation in the neighbourhood of boundaries. The CFD code developer should ensure that the boundary region retains the overall accuracy of the numerical scheme. There is common consent that good practice for outflow boundaries is to set the convective derivative normal to the boundary face equal to zero and to combine this with a streamwise extrapolation of transported quantities. At pressure boundaries the same treatment is usually applied. Open boundaries bring about the following difficulties:

- Non-physical reflection of outgoing information back into the domain, including free surface waves.
- Difficulties in providing information about the properties of the fluid which may inadvertently enter the domain from the outside.
- Difficulties may also arise if open boundary conditions are placed in regions of high swirl, large curvatures or pressure gradients.

Some CFD codes prevent fluid from entering into the domain through open boundaries. In order to avoid undesirable side effects open boundaries should be placed very carefully.

Guidelines

- Ensure that appropriate boundary conditions are available for the case being considered. For swirling flows consult manual to ensure appropriate boundary condition used (for example, radial equilibrium of pressure field instead of constant static pressure). Special non-reflecting boundary conditions are sometimes required for outflow and inflow boundaries where there are strong pressure gradients Giles [1990].
- Check whether the CFD code allows inflow at open boundary conditions. If inflow cannot be avoided at an open boundary then ensure that the transported properties of the incoming fluid including turbulence boundary conditions are properly modelled.

6.10. Application of boundary conditions

In many real applications, there is a frequent difficulty to define some of the boundary conditions at the inlet and outlet of a calculation domain in the detail that is needed for an accurate simulation. A typical example is the specification of the turbulence properties

(turbulence intensity and length scale) at the inlet flow boundary, as these are practically arbitrary in marine CFD. However, for special cases such as propeller or water jet flows the user needs to be aware of these problems and needs to develop a good feel for the certainty or uncertainty of the boundary conditions that are imposed. This can best be achieved if the user knows and understands the application he is calculating.

Additional uncertainties can arise because boundary condition data that needs to be specified is inconsistent with the model being used.

6.10.1. General guidelines on boundary conditions

- Examine the possibilities of moving the domain boundaries to a position where the boundary conditions are more readily identified, are well-posed and can be precisely specified.
- For each class of problem an uncertainty analysis should be carried out in which the boundary conditions are systematically changed within certain limits to see the variation in results. Should any of these variations prove to have a sensitive effect on the simulated results and lead to large changes in the simulation, then it is clearly necessary to obtain more accurate data on the boundary conditions that are specified.

6.10.2. Guidelines on inlet conditions

- Examine the possibilities of moving the domain inlet boundaries to a position where the boundary conditions are easily identified, are well-posed and can be precisely specified.
- For each class of problem a sensitivity analysis should be carried out in which the inlet boundary conditions are systematically changed within certain limits. Aspects that should be examined are:
 - Inlet flow direction and magnitude.
 - Uniform inlet velocity (slug flow) or velocity profile.
 - Variation of physical parameters.
 - Variation of turbulence properties at inlet (see below).

6.10.3. Guidelines on specification of turbulence quantities at an inlet

- A particularly important issue is the specification of the turbulence properties at the inlet to the computational domain and verified quantities should be used as inlet boundary conditions for turbulent kinetic energy k and dissipation ϵ , if these are available as the magnitude can significantly influence the results.
- If there are no data available, then the values need to be specified using sensible engineering assumptions, and the influence of the choice should be examined by sensitivity tests with different simulations.
- For the specification of the turbulent kinetic energy k , values should be used which are appropriate to the application. These values are generally specified through a turbulence intensity level. ERCOFTAC guidelines suggest a variety of values depending on flow type. In hydrodynamics, low "inlet" turbulence levels are likely, but zero turbulence will bring about anomalies in turbulence modelling unless specialised approaches to laminar and transitional regions are adopted.
- The specification of the turbulent length scale, as an equivalent parameter for the dissipation ϵ , is more difficult. For external flows, a value determined from the assumption that the ratio of turbulent and molecular viscosity μ_T/μ is of the order of 10 is appropriate. For simulations in which the near-wall region is modelled, for example in two layer modelling of boundary layers, the length scale should be based on the distance to the wall and be consistent with the internal modelling in the code.
- If more sophisticated distributions of k and ϵ are used these need to be consistent with the velocity profile, so that the production and dissipation term in the turbulence equations are in balance. An inconsistent formulation such as a constant velocity profile and

constant profile of turbulence intensity at the inlet lead to an immediate unrealistic reduction of the turbulence quantities after the inlet. These can be checked by making a plot of the ratio of turbulent to molecular viscosity μ_T / μ . In cases where problems arise the inflow boundary should be moved sufficiently far from the region of interest so that an inlet boundary layer can develop.

- For RSM models the stresses themselves need to be specified, and as these are normally not available an assumption of isotropic flow conditions with zero shear stresses is generally made.

6.10.4. Guidelines on outlet conditions

- The boundary conditions imposed at the outlet should be selected to have a weak influence on the upstream flow. Extreme care is needed when specifying flow velocities and directions on the outlet plane. The most suitable outflow conditions are weak formulations involving specification of static pressure at the outlet plane.
- Particular care should be taken in strongly swirling flows where the pressure distribution on the outlet boundary is strongly influenced by the swirl, and cannot be specified independently of the swirl coming from upstream.
- Be aware of the possibility of inlet flow inadvertently occurring at the outflow boundary, which may lead to difficulties in obtaining a stable solution or even to an incorrect solution. If it is not possible to avoid this by relocating the position of the outlet boundary in the domain, then one possibility to avoid this problem is to restrict the flow area at the outlet, provided that the outflow boundary is not near the region of interest.
- If there are multiple outlets, then either pressure boundary conditions or mass flow specifications can be imposed depending on the known quantities.

6.10.5. Guidelines on solid walls

- Care should be taken that the boundary conditions imposed on solid walls are consistent with both the physical and numerical models used.
- If roughness on the wall is not negligible, significant levels of uncertainty can arise through incorrect specification of roughness within the wall function and when no detailed information is available great care is needed. Research in this area in ship hydrodynamics has been considerable.

6.10.6. Guidelines on symmetry and periodicity planes

- Symmetry and periodicity planes assume that the gradients perpendicular to the plane are either zero (for symmetry) or determined from the flow field (periodicity). If symmetry or periodicity planes cross the inlet or outlet boundaries then care should be taken to specify inlet or outlet variables that are consistent with these.

6.11. Steady flow, symmetry, periodicity, etc.

A symmetric steady computation, or a computation with periodic boundary conditions, is often carried out in order to reduce the computing time and memory required.

There are many applications where the nominal geometry is symmetric but the flow is asymmetric, and the flow field can be asymmetric even in the case of perfect symmetry of the geometry (for example, an oblate spheroid at very high incidence). This can be an important factor in predicting the detail of the dynamical behaviour of fluid flows. The main parameter which gives a preview of the symmetrical behaviour is the Reynolds number. If the Reynolds number is high the flow tends to be asymmetric. This asymmetry can also be forced if the real inflow conditions are not geometrically perfectly symmetrical or some distortions are within the inlet flow.

Due to the physical temporal instability of the flow (e.g. the Karman vortex-street) or due to time-dependent boundary conditions the flow field can be unsteady. This effect should be carefully examined because the flow solvers can often compute a spurious steady solution of

the flow field that is in contradiction to the physics. In cases with very strong unsteady effects within the flow field the solution algorithm does not always converge to a steady solution.

6.11.1. Guidelines

- Check carefully whether the geometry is symmetric or whether a geometrical distortion or disturbance in the inlet conditions is present which can trigger asymmetric solutions.
- Estimate the Reynolds-number of the inflow and check whether the flow could be asymmetric, turbulent and/or unsteady (e.g. by sources or literature).
- After obtaining a steady solution, switch to the transient mode and check whether the solution remains stable.
- If there are difficulties to get a converged steady solution – especially if there is an oscillation of the residuals – switch to the transient mode.
- In case of doubt, the simulation should be unsteady and without symmetry assumptions as boundary conditions.

6.12. Analysis of results, sensitivity studies and dealing with uncertainties

6.12.1. Analysis of results

Most commercial codes come with some kind of post-processing package. This allows many of the flow phenomena to be visualised or plotted in graphical form. The two main steps of post-processing are to determine:

- whether the result is sensible
- whether the result is accurate

Checking the believability of the solution may involve several steps such as checks on conserved variables, visual confirmation that velocities and pressures are smoothly distributed and comparison with other similar problem results. The convergence history will give some indication of whether the problem has reached a steady state solution.

Guidelines:

- Check conserved variables, including an overall force/momentum balance.
- Check that velocities, forces, pressures, etc. have believable values.
- Check whether fluid variables such as velocity and pressure are smoothly distributed over the body and vary rapidly only where expected. Discontinuities may be the result of poor panel definition or insufficient mesh.
- Perform some simple hand calculations to check orders of magnitudes of variables.
- Run simple versions of the problem (e.g. with reduced geometry) to get an idea for the numbers involved.

The accuracy of the result can only truly be determined by knowing the answer in advance. As this is rarely the case the accuracy of the solution will depend on the validation and suitability of the code, the approximations made, the quality of the input parameters and the independent errors (e.g. round-off errors).

Guidelines:

- Ensure that the solution algorithm used is the most suitable, and recognise the approximations used.
- The accuracy of the solution will only be as good as the accuracy of the input conditions.

- Compare the result with similar problems, or simplified versions of the same problem.

6.12.2. Sensitivity studies

Most CFD problems are dependent on mesh quality and resolution, and this may be even more so for free surface problems, such as where the free surface meets the body. It may be easy to find in the literature a suggested panel or mesh density for a particular problem but it is important to examine the sensitivity of variables such as these on the solution. This may also take an iterative form, where the initial solution has highlighted an area of insufficient mesh resolution or skewed panels or grid that need improvement.

Guidelines:

- Perform the calculation using several different panel and grid densities
- Investigate the sensitivity of boundary conditions
- If time permits run the problem using a different source code and compare the results
- Investigate the effects of different viscous approximations or turbulence models

6.12.3. Dealing with uncertainties

As described in section 3.2, uncertainties arise through lack of knowledge. This can be a lack of knowledge of the details of the problem to be modelled, or of the methods and approximations used to solve the problem. The latter can only be solved by increased user awareness to the theories and methods used. Uncertainties can also occur because of simplification of the problem due to modelling constraints.

Guidelines:

- Avoid over-simplification of the model which may omit important effects
- Be aware of the magnitude and implication of errors (e.g. round-off errors)
- Scale factor is important – solution is much easier at model scale (smaller Reynolds number) but there may be difficulties scaling up the results, i.e. Froude / Reynolds scaling differences.

7. Application examples

The following application examples are given in order to illustrate some of the issues described in these guidelines.

7.1. Example of Wave Pattern Calculations for Steady Ship Flow

Application example calculated with SHIPFLOW/XPAN supplied by FLOWTECH International AB, Gothenburg, Sweden.

7.1.1. Introduction

The purpose of the example is to show a computation of the wave pattern generated by a commercial vessel known as "Ville de Mercure" or "The Hamburg test case". A potential flow code of Rankine source type is used to solve the problem. The computed waves are compared to experiments along longitudinal wave cuts. The experimental data can be found in the MARNET data base.

7.1.2. Geometry and boundary conditions

The main dimensions of the hull are $L_{pp}=153.68\text{m}$, $B=27.5\text{m}$, $T_f=9.2\text{m}$, $T_a=10.3\text{m}$. The geometry of the hull is described by a set of offset points as shown in figure 1. In total there are approximately 120 stations having about 40 points each. The hull is divided into 4 parts, main hull, stern, fore bulb and aft bulb. No appendages are included and the transom stern and the stern bulb are left open.

The free surface extends from half a shiplength upstream to one shiplength downstream of the hull in the longitudinal direction and to 0.8 shiplengths in the transverse direction in order to capture the Kelvin angle within the downstream boundary. This size of the free surface is sufficient to compute the near field waves around the hull.

A condition of zero flow in the surface normal direction is applied on the hull and the kinematic and dynamic free surface boundary conditions are applied on the free surface.

The computations are performed for the Froude number 0.2385.

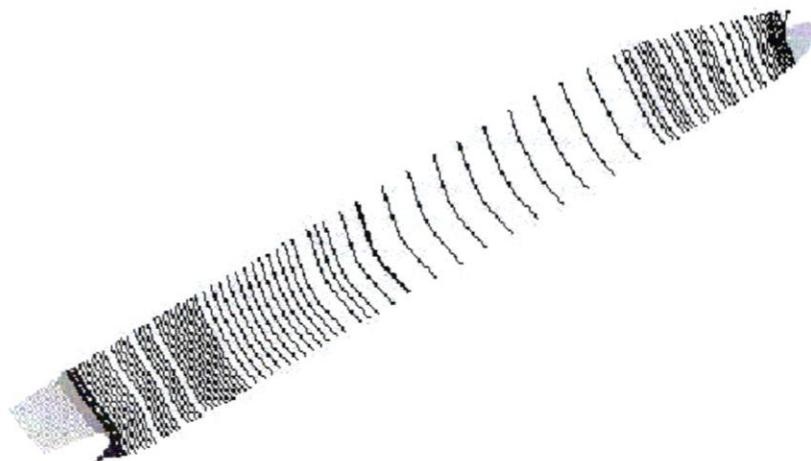


Figure 7.1 Hull offsets

The free surface is divided into a main part and a transom part behind the transom stern. 30 panels are used per fundamental wave length in the longitudinal direction. In total 30 panels are used in the transverse direction with a clustering towards the waterline. The size of the first panel at the waterline is $0.015 \cdot L_{pp}$. A hyperbolic tangent stretch function is used in the transverse direction. At the average the expansion factor is 1.04 which is considered small enough to capture the main contribution to the diverging waves (Prins, Raven 1997).

Figure 7.2. Panel distribution on the hull

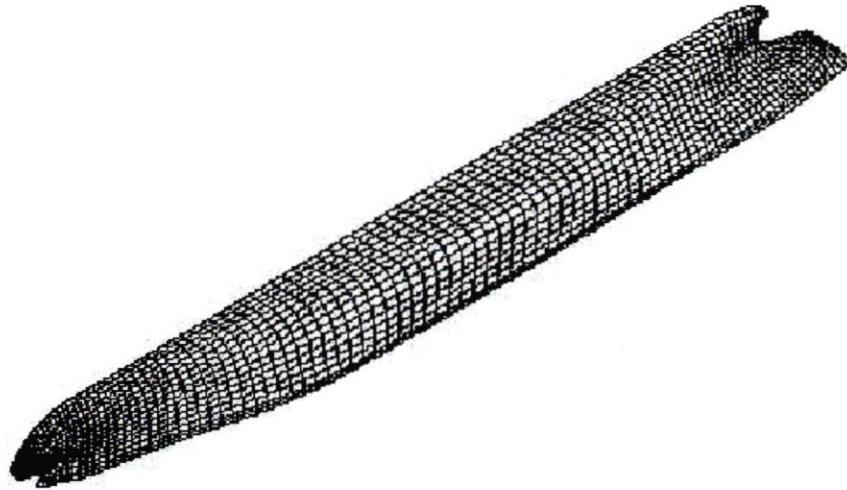


Table 7.1 Number of panels on the hull

Number of panels	Size of first panel	Size of last panel	Number of panels	Direction
80	$0.005 \cdot L_{pp}$	$0.0075 \cdot L_{pp}$	17, uniform	longitudinal
8			9, uniform	stern
9			16, uniform	fore bulb
4			4, uniform	stern bulb

As mentioned above the hull is divided into 4 groups. Each part is panelized individually using a structured mesh. The number of panels are selected to give a good representation of both the geometry and the flow field. On the main part of the hull the number of panels in the longitudinal direction is selected to resolve the wavy behavior of the flow along the waterline. The number of panels is specified in table 1 for the four groups and the panel distribution is shown in figure 7.2.

7.1.3. Grid

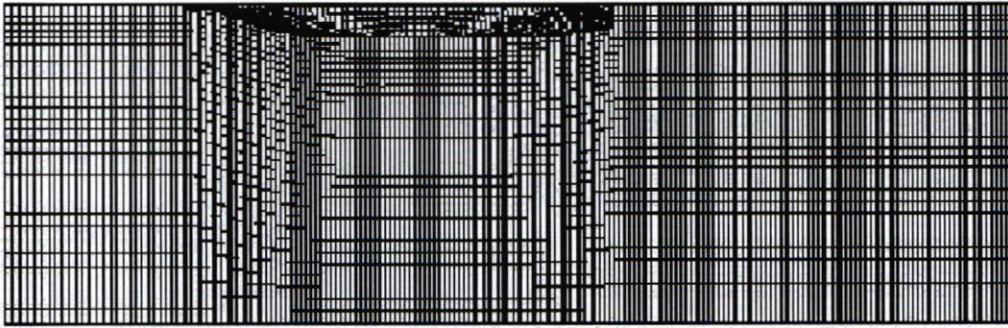


Figure 7.3. Free surface panels

7.1.4. Features of the simulation

Only a brief description of the numerical method for solving the free surface problem is included below. Details of the method can be found in (Janson, C-E. 1997).

A potential-flow method is used to solve the free surface problem. Steady state incompressible flow in a coordinate system that moves with the body is also assumed in the present formulation.

The free-surface problem is non-linear since the free-surface boundary conditions are non-linear and must be satisfied on the initially unknown wavy free surface. The solution method for the non-linear problem used in the present method is to linearise the free-surface boundary condition around a known base solution and to solve the problem in an iterative manner. In each iteration the problem is linearised with respect to the solution from the previous iteration and the first iteration is started from a base flow that may be the undisturbed flow or a zero Froude number flow where a Neumann condition is applied on the free-surface. The linearised free surface boundary conditions are in the first linear solution applied on the undisturbed free surface and are in the following iterations moved to the wavy free-surface computed in the previous iteration. In each iteration the wave height is computed from the linearised dynamic free surface boundary condition.

The hull surface and the free surface are discretised using a large number of quadrilateral Rankine source panels. The hull panels are assumed to be parabolic having a linearly varying source strength and a boundary condition of zero flow through the panel is applied at the panel control point. The free-surface panels are flat and have a constant source strength. The combined free-surface boundary condition applied at the control point on each panel, includes velocity derivatives. In the present method the velocity derivatives are calculated using an upwind finite difference operator. The choice of difference operator is very important for the performance of the method since both the damping (amplitude error) and dispersion (wave length error) are influenced by the finite difference operator. Also the necessary condition of no waves propagating upstream (radiation condition) is introduced by the use of an appropriate finite difference operator. In the present method a four-point operator is used. The free-surface source panels are raised a small distance above the free-surface level and the free-surface control points are shifted a small distance upstream in order to reduce the damping and the wavelength error that appears for the original four point operator. A central difference operator is used to compute the velocity derivatives in the transverse direction.

7.1.5. Results

A contour plot of the computed wave pattern is shown in figure 7.4 and three longitudinal wave cuts are compared to experimental data in figures 7.5 – 7.7. The bow wave is well predicted as can be seen in the figures while the stern wave is over predicted.

A grid dependence study was also carried out using 15, 20, 25 and 30 panels per fundamental wave length in the longitudinal direction. The hull panelization and the transverse distribution of free surface panels was not changed during the grid dependence study. The longitudinal wave cuts were compared for the four panelizations and a large difference was noted between 15, 20 and 25 panels while the difference was small between 25 and 30 panels. It was therefore concluded that 25 - 30 panels per fundamental wave length is enough to resolve the wave pattern.

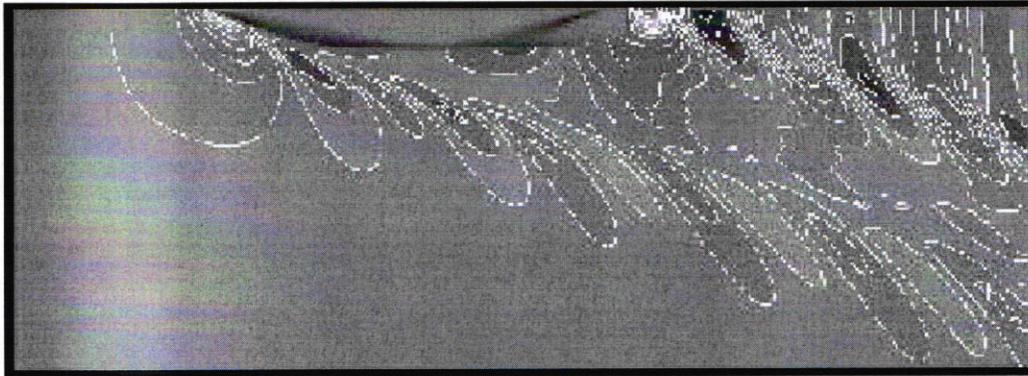


Figure 7.4. Computed wave pattern

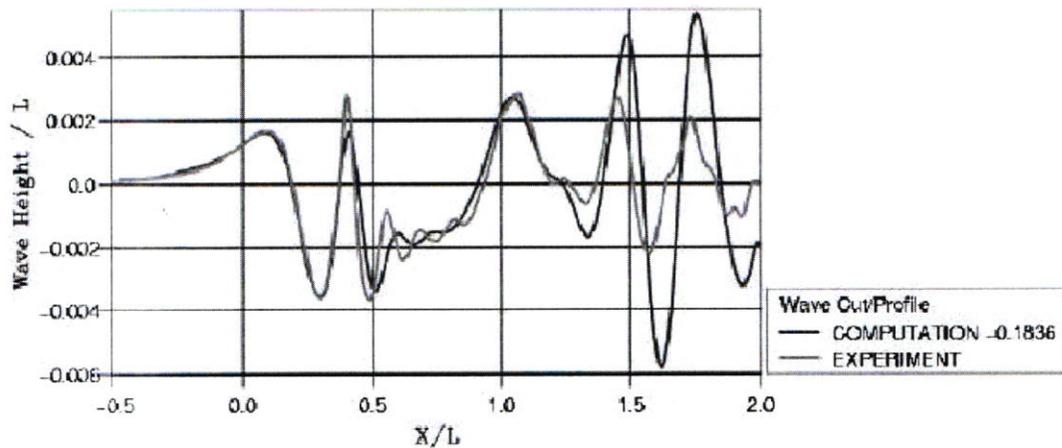


Figure 7.5. Wave profile, longitudinal cut at $y=0.1836 \cdot L_{pp}$, comparison to experiments

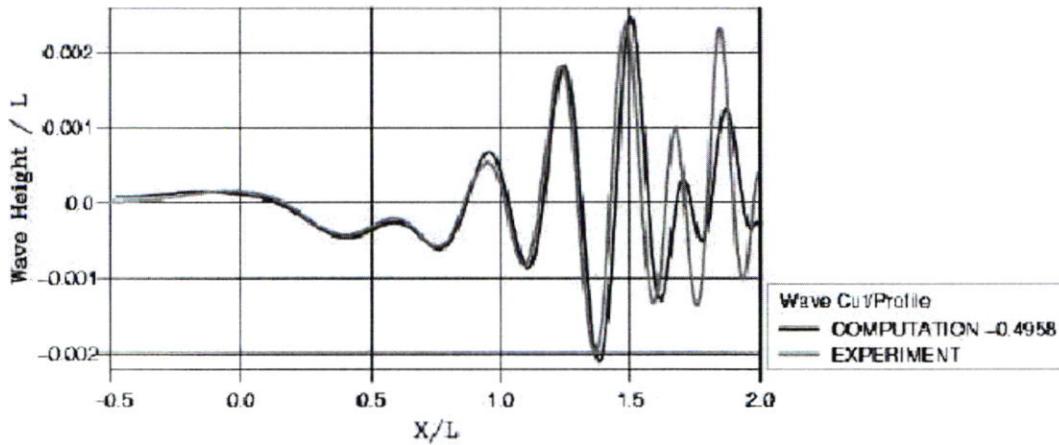


Figure 7.6. Wave profile, longitudinal cut at $y=0.4958 \cdot L_{pp}$, comparison to experiments

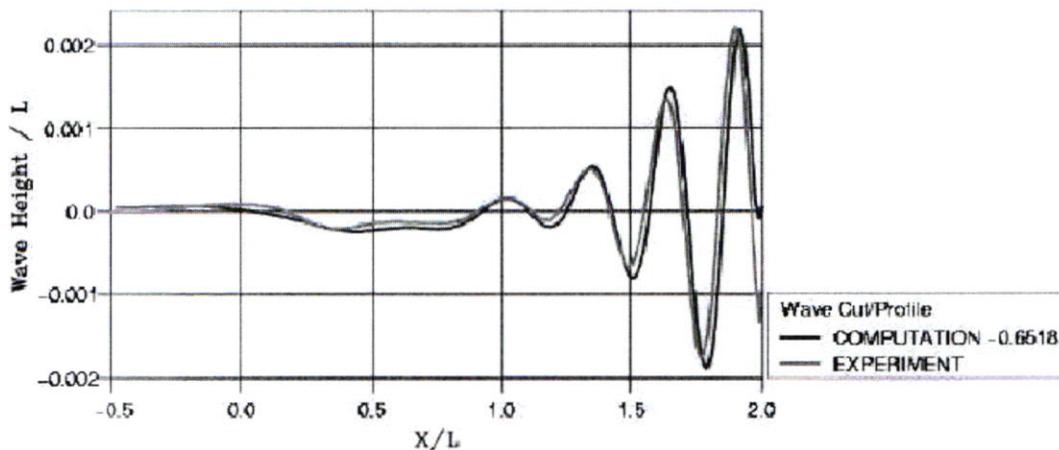


Figure 7.7. Wave profile, longitudinal cut at $y=0.6518 \cdot L_{pp}$, comparison to experiments

7.1.6. Conclusions

The bow wave and the diverging waves from the fore part of the hull can be well predicted using a potential flow method. The stern wave is however too large which is due to the neglected viscous effects of the boundary layer displacement and also due to the potential flow representation of the flow just behind the submerged transom stern. A computation including viscous effects is necessary to improve the prediction of the stern wave.

7.1.7. References

1. Janson, C-E. 1997 Potential Flow Panel Methods for the Calculation of Free-surface Flows with Lift. Department of Naval Architecture and Ocean Engineering, Phd thesis, Chalmers University of Technology, Gothenburg, Sweden.
2. Prins, H.J., Raven H.C. 1997 Improving the RAPID resistance prediction, CALYPSO report, Task 2.2, Deliverable D2.2, Brite/Euram III BE95-1721.

7.2. Example of viscous stern flow calculations

Application example calculated with SHIPFLOW/XVISC supplied by FLOWTECH International AB, Gothenburg, Sweden.

7.2.1. Introduction

The purpose of this example is to show a computation of the flow around the stern for a ship known as "Ville de Mercure" or "The Hamburg test case". A single block RANSE solver is used for the computation. The computed wake is compared to experiments available in the MARNET data base.

7.2.2. Geometry and boundary conditions

The main dimensions of the hull are $L_{pp}=153.68\text{m}$, $B=27.5\text{ m}$, $T_f=9.2\text{ m}$ $T_a=10.3\text{ m}$. The geometry of the hull is described by a set of offset points as shown in figure 1. In total there are approximately 120 stations having about 40 points each. The hull is divided into 4 parts, main hull, stern, fore bulb and aft bulb. No appendages are included and the stern bulb is extended and closed.

The computational grid starts midships and ends a quarter of a shiplength downstream of the stern. The radius to the outer cylindrical boundary is 0.4 shiplengths.

Boundary conditions at the inlet plane:

Velocity components are extracted from a boundary layer computation and outside the boundary layer from a potential flow solution. The turbulent quantities are computed from analytical formulas based on the velocity profile in the inlet plane. In addition the equations for the turbulent quantities are solved assuming a zero velocity gradient in the main flow direction. This second step is used to obtain a smooth distribution of the turbulent quantities.

Boundary conditions at the outer boundary:

The tangential velocity components and the pressure are obtained from a potential flow solution and the velocity component normal to the boundary is computed from the continuity equation. A zero normal derivative is assumed for the turbulent quantities.

Boundary conditions at the flat free surface and the center plane:

A symmetry condition is assumed on both planes.

Boundary conditions at the outlet plane:

The second derivative of the velocity components and the turbulent quantities are assumed to be zero in the main flow direction. The pressure is set to zero.

Boundary conditions on the hull:

No slip and a wall law is used on the hull surface.

The computations are performed for the Reynolds number $1.28 \cdot 10^7$.

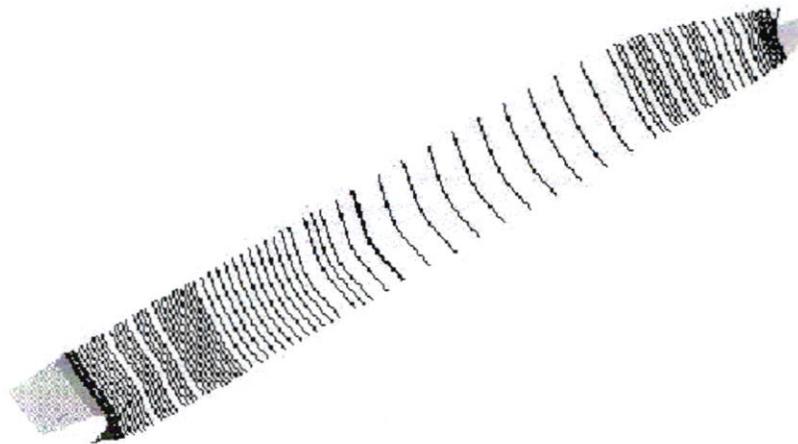


Figure 7.8 Hull offsets

Grid

As mentioned above the hull is divided into four groups, three of them are used to generate a structured single block grid around the stern. Grid points are distributed on the boundaries and between the boundaries to form an initial volume grid. The grid points are clustered towards the hull in order to generate a grid that can be used together with the wall law (y^+ about 50 for the first point outside the hull). A Poisson solver is then used to make the grid as orthogonal as possible. The grid points are allowed to move along the boundaries during this process. Figures 7.9 – 7.11 show a grid where 120 points were used in the longitudinal direction, 40 points in the girthwise direction and 50 points in the radial direction.

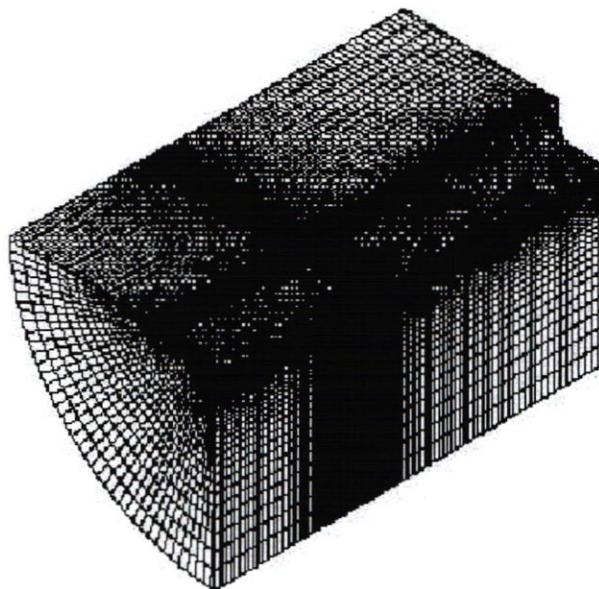


Figure 7.9. Single block grid for the stern flow computation

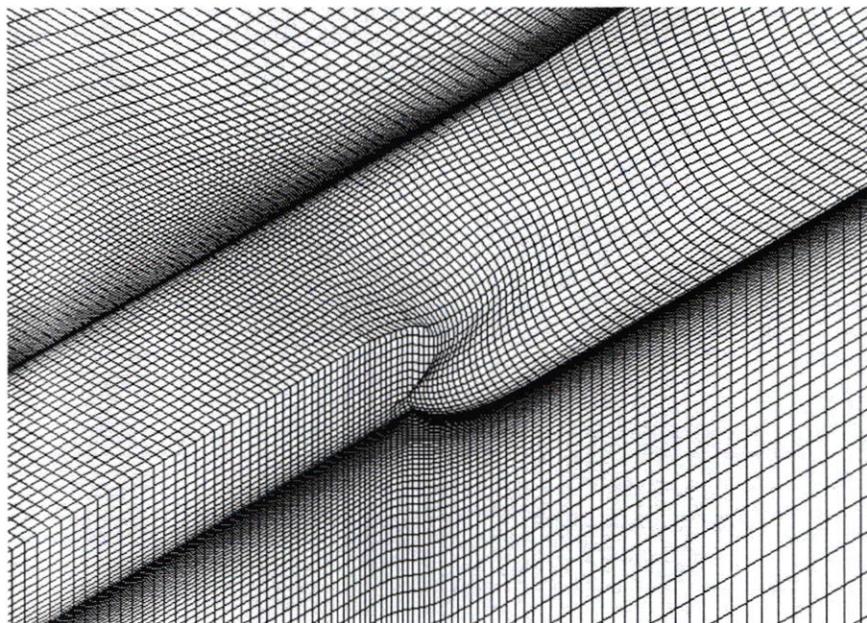


Figure 7.10. Details of the grid at the stern

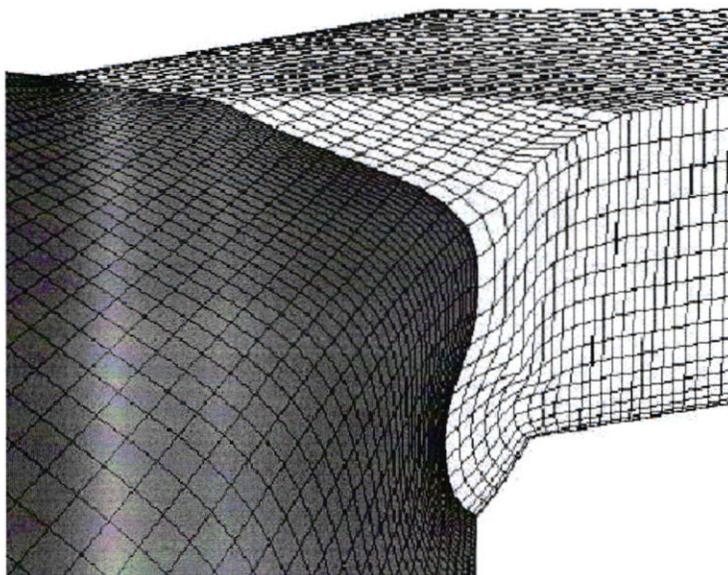


Figure 7.11 Details of the surface grid at the stern bulb.

7.2.3. Features of the simulation

The time averaged Navier-Stokes equations for incompressible flow are solved. A predominant flow direction is assumed for the flow around the hull and a simplified set of equations can be used, assuming that the stress derivative in this direction is small. The equations for the velocities then becomes parabolic and a marching technique can be used for the solution. The pressure calculation is, however, elliptic and the method can be referred to as partially parabolic. A curvi-linear non-orthogonal coordinate system is used and both the independent and the dependent variables are transformed to this system.

Transport equations are solved for the turbulent kinetic energy and its rate of dissipation. A wall law represents the velocity distribution close to the hull surface.

Numerically the problem is solved using a finite-difference method. In the cross-plane a finite-analytic scheme is used while a second order upwind scheme is used in predominant direction. The pressure-velocity coupling is based on the SIMPLER algorithm.

The theory for the Navier-Stokes method is described in detail in (Broberg, L. 1988), (Larsson, L., et al. 1989) and (Ohkusu, M., ed. 1996).

7.2.4. Results

The computed and measured wake are compared in figures 7.12 and 7.13. The general behaviour of the wake is well captured in the computations but some details of the iso-wakes are missing in the inner part of the boundary layer, in particular at $x=0.9675 \cdot L_{pp}$.

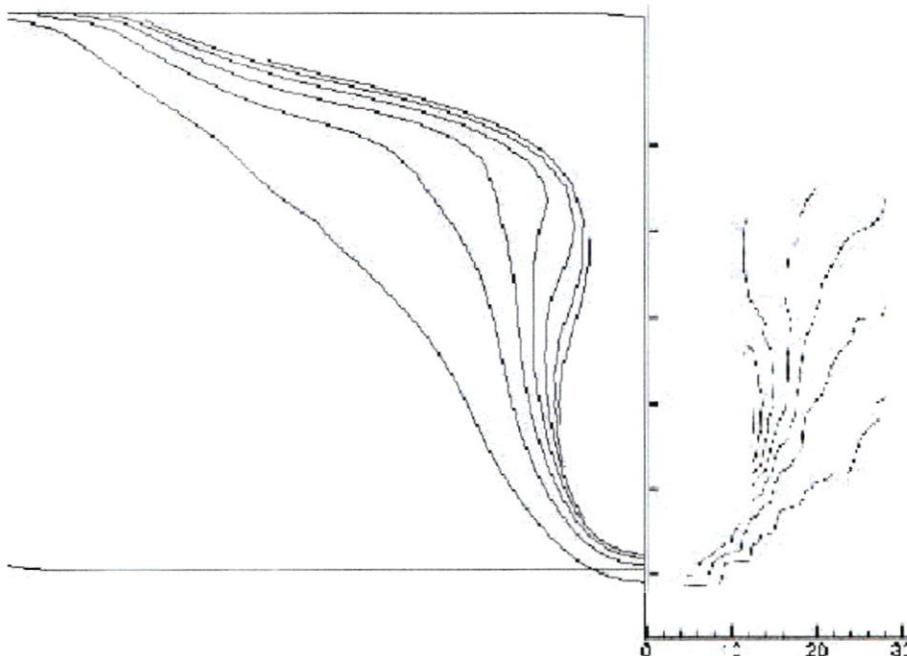


Figure 7.12. Comparison of the computed (left) and the measured (right) wake at $x=0.9379 \cdot L_{pp}$

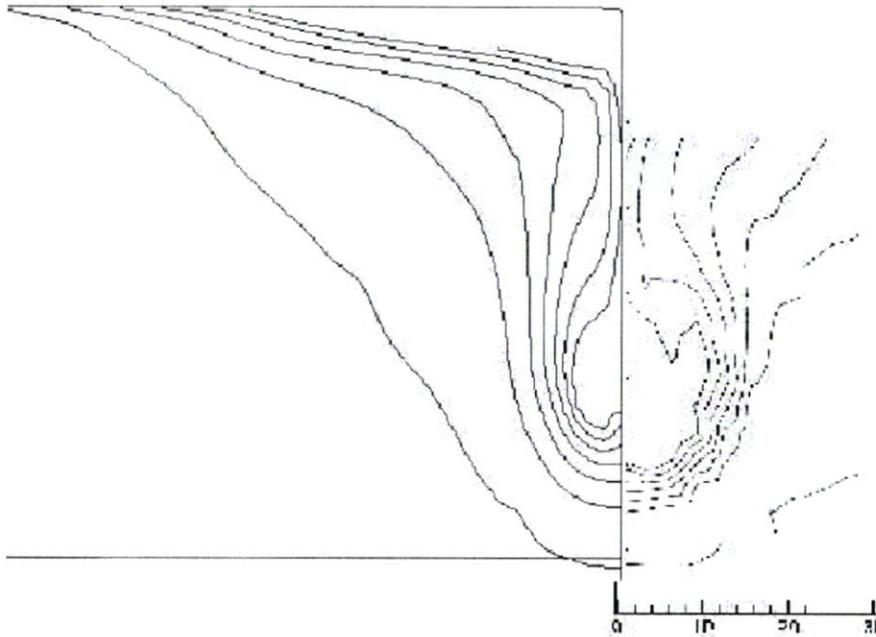


Figure 7.13. Comparison of the computed (left) and the measured (right) wake at $x=0.9675 \cdot L_{pp}$

7.2.5. References

1. Broberg, L. 1988 Numerical Calculation of Ship Stern Flow. PhD Thesis, Department of Mechanics, Chalmers University of Technology, Gothenburg, Sweden.
2. Larsson, L., Broberg, L., Kim, K. J. and Zhang, D. H. 1989 New Viscous and Inviscid CFD Techniques for Ship Flows. 5th International Conference on Numerical Ship Hydrodynamics, Hiroshima, Japan.
3. Ohkusu, M., ed. 1996 Advances in Marine Hydrodynamics. Computational Mechanics Publications, Southampton, Boston, pp. 1-75.

7.3. Example of Unsteady Manoeuvring Calculations

Application example provided by Sirehna, 1, Rue de la Noe, Nantes.

7.3.1. Introduction

Numerical simulations of three-dimensional unsteady viscous free surface flow past a ship in drift and in rotating motion using the Reynolds Averaged Navier-Stokes Equations are presented. A fully coupled method for the velocities, pressure and free surface elevation discrete unknowns is used. The purpose of this work is to generalise algorithms while preserving their efficiency in order to take into account non-symmetrical free surface flows.

The areas of best practice that are illustrated by this example are referred to in sections 2.5.2, 2.6.2.1, and 6.2.2.4.

The whole solution of non-symmetrical ship flows requires to compute three dimensional unsteady turbulent boundary layers with flow separation connected to complex free surface effects : Reynolds Averaged Navier-Stokes Equations written under convective form in an unsteady curvilinear computational space fitted at each iteration to the hull and to the free surface are used. Fully non linear free surface conditions are solved using an efficient fully coupled algorithm and turbulence effects are taking into account through classical $k-\omega$ modelisation.

Series 60 experiments (free surface elevation along the hull, C_{fx} , C_{fy} , C_{mz}) with attack angle $\theta=5^\circ$ due to J. Longo and F. Stern from Iowa University provide the experimental data for comparison. No experiment results are available for the gyration case.

7.3.2. Geometry and boundary conditions

The hull form used in these calculations is that of the standard Series 60 hull, block coefficient = 0.6. The computational domain is hemispherical, with a radius equal to five times the water line length. The computational domain is divided into the port and starboard halves with an overlapping multi-block strategy used to deal with the interface.

The fluid is modelled as a viscous, incompressible and Newtonian. The flow is treated as unsteady and utilises a $k-\omega$ turbulence model. A wave-breaking criterion based on free surface curvature is used. This criterion, published by Subramani et al., fixes the limiting value of $|kh|$ where k is the free surface curvature and h the free surface elevation for a wave not to break : $|kh| < 0.5$.

For non-symmetrical free surface flows, topological boundaries do not coincide with physical boundary conditions. Overlapping techniques have been left to joined boundaries requiring the development of specific discrete operators and new linear system solvers allowing the discretization molecules to cross topological boundaries.

Free surface boundary conditions are one kinematic condition, two tangential dynamic conditions and one normal dynamic condition, with further details given in reference 1.

7.3.3. Grid

Structured curvilinear grid (O-O topology) fitted to the hull and the free surface has been used in this example.

The finest grid level has $2 \times 89 \times 73 \times 33 = 428,802$ nodes and the coarsest grid has $2 \times 57 \times 49 \times 33 = 184,338$ nodes. The $k-\omega$ turbulence model is used without wall function which requires a great concentration near the hull with the first grid point located at $s/l = 10^{-5}$, where s is the curvilinear co-ordinate normal to the hull.

Typically 3 grids are used to assess the convergence. In practice, all of these grids ensure a good accuracy of the hull integrated data (forces). The final mesh has been chosen so that the calculated flow is converged up to about 0.5 ship length.

The best practice guidelines described in section 6.8 should be referenced

In the drift angle case and the gyration case, calculation are performed for model scale at a Reynolds number $Rn = Ua.l/\nu = 5.3 \cdot 10^6$, a Froude number $Fn = Ua/\sqrt{gl} = 0.316$ and a Bond number which traduces surface tension effects $Bn = \rho g l^2/\gamma = 1.3 \cdot 10^6$, where l is the boat length.

For the gyration case, the rotation velocity Ω (in radians/s) is choose according to $\Omega/Ua = 8.73 \cdot 10^{-2}$ involving a curvature radius $Rg/l = Ua/(\Omega l) = 11.5$. The rotation axis is vertical including point $R(0, -Rg, 0)$ and the boat is fixed in all the degrees of freedom.

Calculations are performed during 500 time iterations with a non-dimensional time step $\tau = Ua.\delta t/l = 0.025$ for all grids.

Calculations simulate exactly a towing tank test in that, during a first stage, the hull is in uniform acceleration up to the nominal velocity and then velocity is held constant thereafter.

A second order (in space and time) implicit fully coupled finite difference scheme has been used in the calculation. Convection terms are computed using an upwind second order scheme that needs a 13 nodes cell. The diffusion terms need 7 nodes for second order derivatives and 12 nodes to express cross second order derivatives while pressure gradient requires 8 nodes for each component. Pressure equation uses Rhie and Chow method (27 nodes).

7.3.4. Results

General agreement is satisfactory (drift angle case). The bow and stern local free surface on pressure side or suction side has good amplitude and computed and experimental wave patterns are very similar. Nevertheless the computed wave-field presents a typical current fault of most three-dimensional free surface viscous flow calculations, in that the wave amplitudes appear to be damped in the far-field.

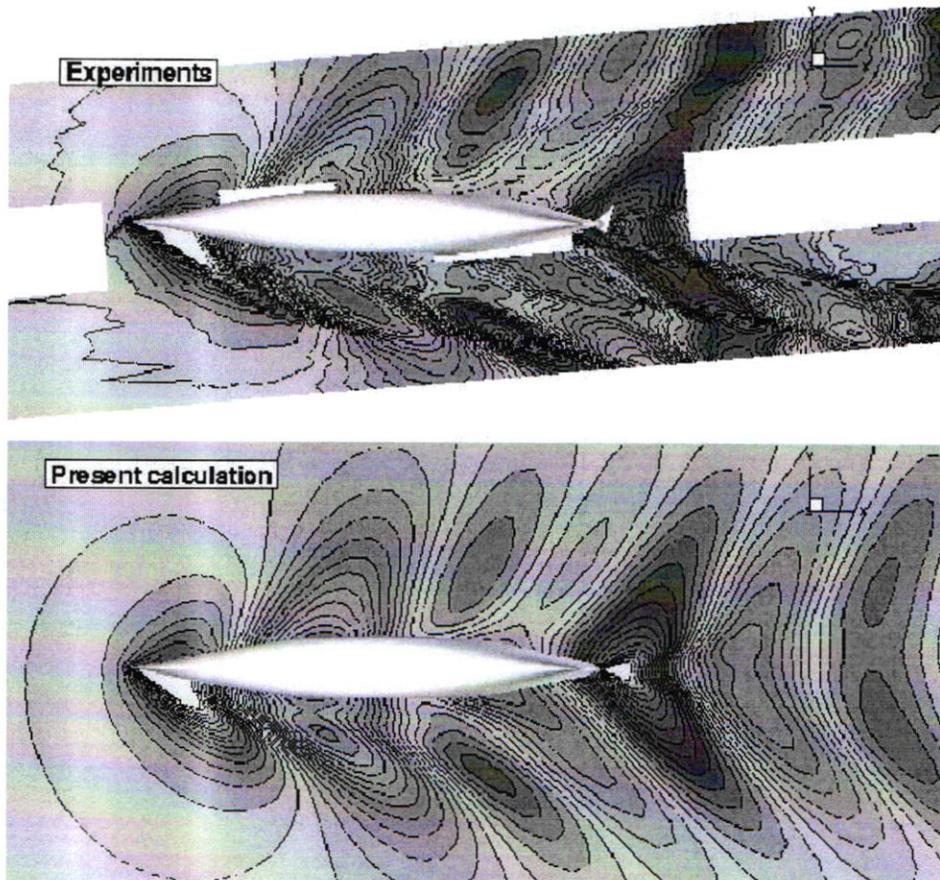


Figure 7.14 : wave field for $Fn = 0.316$ and $Rn = 5.3 \cdot 10^6$

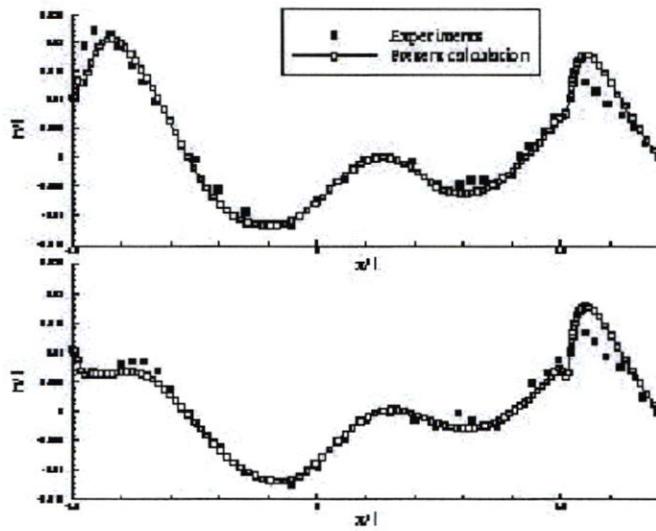


Figure 7.15 Free surface elevation along the hull (pressure side on the top and suction side on the bottom)

	C_{fx}	C_{fy}	C_{mz}
Experiments	$6.6 \cdot 10^{-3}$	$10.6 \cdot 10^{-3}$	$-0.73 \cdot 10^{-3}$
Calculation	$6.4 \cdot 10^{-3}$	$9.3 \cdot 10^{-3}$	$-0.72 \cdot 10^{-3}$

Table 7.2 : Results for the drift angle case.

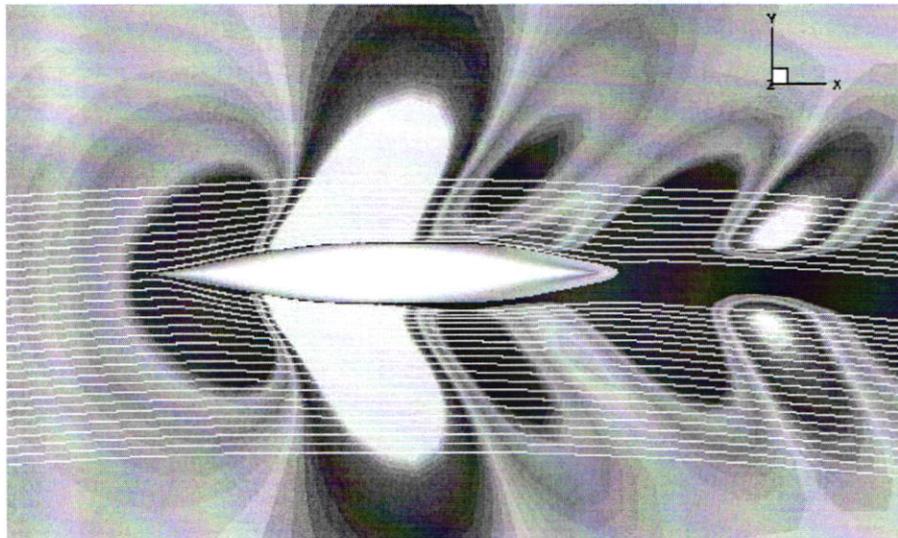


Figure 7.16 : modulus of velocity and streamlines on the free surface for $Rg/l = 11.5$

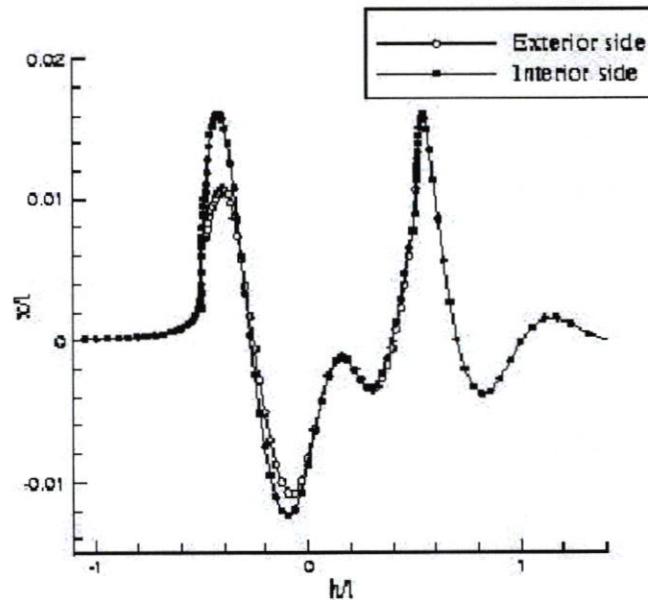


Figure 7.17 Free surface elevation on both sides of the hull for $Rg/l = 11.5$

	C_{fx}	C_{fy}	C_{mz}
Calculation ($Rg/l = 11.5$)	$6.1 \cdot 10^{-3}$	$1.4 \cdot 10^{-5}$	$-0.31 \cdot 10^{-3}$
Calculation ($Rg/l = +\infty$)	$5.7 \cdot 10^{-3}$	0	0
Experiments ($Rg/l = +\infty$)	$5.9 \cdot 10^{-3}$	0	0

Table 7.3 : Results for the gyration case.

7.3.5. Identification of errors and uncertainties

Model errors of interest in this case are those classically associated with Newtonian fluid and choice of turbulence model. The wave breaking model may also have some influence, but this is expected to be of relevance only to the near field solution.

With regard to numerical errors, second order numerical scheme is used in both time and space in this case, minimising the potential for numerical diffusion to orders higher than the physical diffusion processes as recommended.

With regard to convergence errors, a very good convergence is ensured thanks to the use of a coupled method

User errors are likely to be low relative to general purpose CFD applications in industry owing to the automatic approaches used in grid generation and the reduced number of parameters involved in the method.

7.3.6. Conclusions

Classical grid over-lapping techniques coming from multiblock solvers have been tested in order to transmit flow information (mass and momentum conservation) through topological (but not physical) boundaries joining the two blocks (starboard and port). Unfortunately this method gives very slow convergence rate that suppress the benefit of fully coupled techniques. Better way consisting in coding new schemes on and near the topological boundary and new specific linear solvers is developed here. In this case, the two parts of the

grid are not overlapped and convergence rate in the whole domain appears better than convergence with symmetrical flow.

This method has been validated in two important cases for hydrodynamics point of view: ship moving with a non-zero attack angle and ship in rotating motion. In both case non-symmetrical flows and continuity of dependant unknowns crossing boundaries are shown.

Concerning drift simulation general wave pattern shows a good agreement with experiments particularly on the bow wave where amplitude on the suction and the pressure sides are good. Nevertheless important wave damping can be observed at a distance from the hull as usual using RANSE solver. Converged values of resistance coefficients are in very good accordance with experimental values.

Concerning gyration simulation, experimental values concerning local variables (pressure, velocity, free surface elevation) do not exist today and validations are not very easy. Nevertheless an increase of resistance can be observed comparing with symmetrical flow simulation.

7.3.7. References

1. B. Alessandrini, G. Delhommeau (Ecole Centrale de Nantes, France)
"Viscous free surface flow past a ship in drift and in rotating motion", 22nd Symposium on Naval Hydrodynamics, Washington D.C., August 1998.
2. B. Alessandrini (Ecole Centrale de Nantes, France)
"Etude numérique de l'écoulement d'un fluide visqueux autour d'une carène de navire en incidence et en giration forcée", July 1997.

7.4. Example of propeller flow calculations

Application example provided by VTT, with thanks and acknowledgements to:

Moustafa Abdel-Maksoud, Potsdam Model Basin, Marquardter Chaussee 100, D-14469 Potsdam. **Florian Menter**, AEA Technology GmbH, Staudenfeldweg 12, D-83624 Otterfing. **Hans Wuttke**, Potsdam Model Basin, Marquardter Chaussee 100, D-14469 Potsdam.

7.4.1. Introduction

This example demonstrates the manner in which the solution for flow around a marine propeller can be achieved. It concentrates on issues of grid generation, and uses a general purpose CFD code to provide the core solver. Since the code (CFX-TASCflow) uses hexahedral grids, the difficulties to be overcome to match this grid topology to the geometry of a skewed propeller and shaft are considerable. Some compromises on the best practice guidelines described earlier are therefore inevitable.

7.4.2. Geometry

The propeller series 4021 of SVA, (Heinke et al, 1993) is used in this example, being a suitable compromise between a conventional and a highly skewed geometry. The geometry is shown in the following figure. Note that full details of the geometry should be obtained from HSVA.

In defining the extent of the computational domain, it is assumed that the axis of the propeller and the direction of the flow coincide. All blades of the propeller are assumed identical. As a result, the flow is periodic with respect to the blades and only one blade has to be considered. Periodic boundary conditions have to be applied in circumferential direction. The RANS solver features a general grid interface (GGI) capability, which allows to join grid blocks with non-matching node distributions. This feature can be applied to the periodic boundaries encountered in the simulation of ship propellers. Therefore, the grid distribution on the two periodic boundaries in circumferential direction does not have to match, leading to a significant simplification of the grid generation procedure. However, the present method has also been applied to generate grids with matching nodes at the periodic boundaries. The periodic boundaries are placed between the blades.

7.4.3. Grid

One of the main obstacles in computing ship propeller flows by solving the Navier-Stokes equations is the complexity involved in the generation of suitable grids. Compared to other lifting bodies, like wings on airplanes, there are additional difficulties associated with the geometry of a propeller:

- Periodicity in circumferential direction,
- Strong twisting of the blade central plane,
- Complex shape of modern propellers,
- Stagnation point on hub close to propeller,
- Limited space for grid generation behind the ship.

The grid generation is based on a commercial grid generation package employing block-structured hexahedral grids. It can be fully parameterised, i.e. templates can be written which are independent of the geometry. Different geometry shapes can then be substituted

and a grid can be generated automatically. The grid can be optimised by moving topological points on geometric lines or surfaces.

As the grid generation is based on a block-structured hexahedral grid, it is necessary to develop a topology for the arrangement of the different blocks. Fig. 7.18 gives a general overview over the positioning of the grid around the propeller blade. The grid is wrapped around the blade, following approximately the angle of the blade against the propeller axis. Away from the blade (in axial direction), the grid straightens out and runs parallel to the axis.

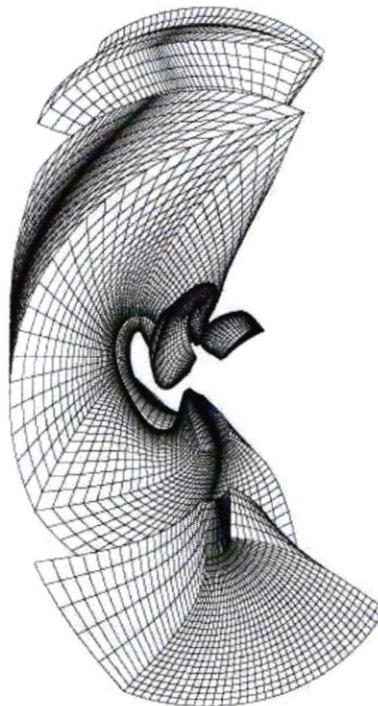


Figure 7.18 General overview of the propeller grid structure

Figure 7.19 below illustrates the distribution of the developed grid projected on to the blade surface.

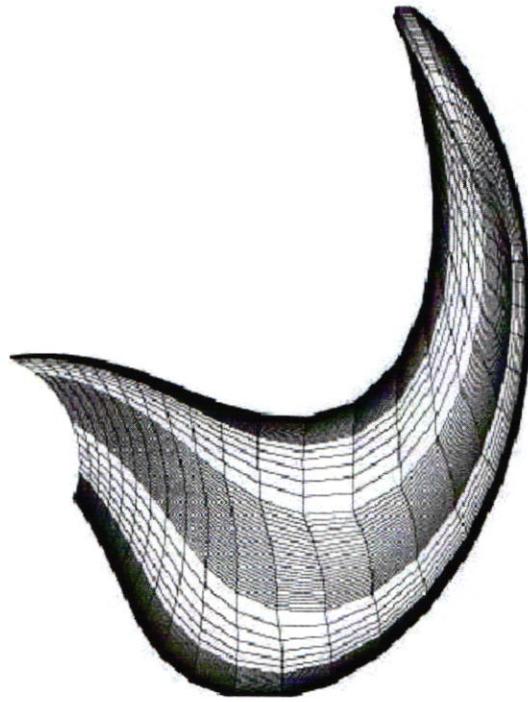


Figure 7.19: Projection of grid on to the blade surface.

To resolve the strong pressure gradients in the leading edge region (stagnation line and suction peak), the grid is locally refined in that area (Fig. 7.20 below).

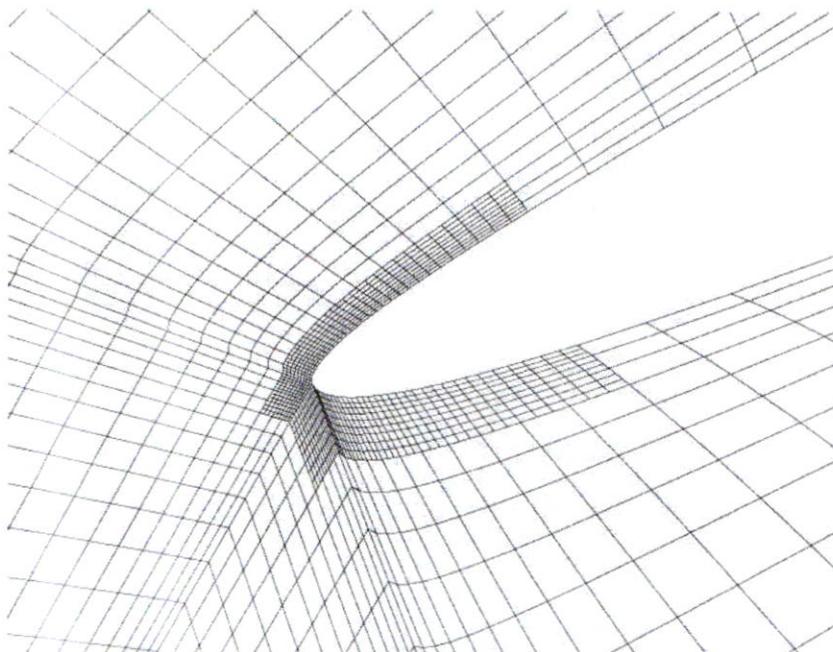


Figure 7.20 Local grid refinement detail at the blade leading edge.

The quality of a hexahedral grid for the simulation of fluid flow can be measured by a number of parameters. Most important are the grid angles. The optimum grid has 90 degree angles everywhere because then, the numerical accuracy and the robustness of the discretisation scheme are best. However, orthogonal grids can only be achieved for simple geometries. In most applications, the grid contains angles of 45 degrees and less.

For a complex geometry like a high-skew marine propeller, angles of around 20 degrees and smaller can hardly be avoided. The analysis of distribution of minimal grid angles per cell face for the grid of the propeller 4021 shows that is only 0.3% of grid cells with angles of around 20 degrees. For the propeller 2133 the smallest angles are around 10 degrees as a result of the higher skew. Most cells however are around 45 degrees, as has to be expected from the underlying topology.

7.4.4. Features of the Simulation

7.4.4.1. Equations

The flow around ship propellers is computed in a rotating co-ordinate system attached to the propeller. The RANS equations in a rotating co-ordinate system involve additional terms compared to those in an inertial system. It is vital that users of commercial, general purpose, CFD codes check that these additional terms are included where rotating co-ordinate system options are offered. Details of the modified equations can be found in the references given later.

The computations have been performed with a RANS solver based on a conservative, second order accurate, finite volume scheme with collocated variables. An algebraic multi-grid algorithm is employed to accelerate the convergence of the linear solver.

The effect of turbulence is modeled using the standard $k-\epsilon$ model (Launder and Sharma, 1974). Wall function boundary conditions are used. However, the grid generation procedure could also provide grids for the resolution of the viscous sub-layer.

7.4.4.2. Boundary conditions

The following boundary conditions were specified:

- Inlet: Velocity vector and k and ϵ specified,
- Outlet: Pressure specified at one face. All other variables extrapolated with zero gradient along grid line,
- Propeller and hub: Wall function boundary conditions,
- Tunnel walls: Slip condition,
- Side boundaries: Periodicity for all variables.

7.4.4.3. Flow conditions

The propeller 4021 has been tested experimentally over a wide range of operating conditions in a closed water tunnel (Heinke et al, 1993). The propeller model has an outer diameter $D = 0.25$ m and a hub diameter $d = 0.07$ m. The test-section was rectangular 0.85×0.854 and has been approximated by a cylinder of diameter $D_t = 0.62$ m. Off-design conditions were achieved by changing the speed of the water in the tunnel.

7.4.5. Results

The numerical results agree very well with the experimental data for the propeller 4021. Some differences appear at the extreme off-design conditions. There is little influence of the grid density on the predicted performance characteristics of the propeller for the cases covered here.

Numerical results for the axial velocity profiles at 0.17 D behind the propeller 4021 are shown for the standard grid (250,000 nodes) and the fine grid (2,107,596 nodes). For all four radial locations, the overall level and the general shape of the velocity profiles agree well with the experimental data. Differences in the details of the wake are most likely due to deficiencies in the turbulence model or insufficient resolution. The differences in the computed results on the two grids indicate that grid independence was not achieved on the standard grid. Figure 7.21 shows the computed results in terms of the non-dimensional thrust and torque,

$$K_T = \frac{N_b \int F_z dA}{\rho N^2 D^4} \quad (27)$$

$$K_Q = \frac{N_b \int (F_x y - F_y x) dA}{\rho N^2 D^5} \quad (28)$$

and efficiency:

$$\eta = \frac{K_T W_z}{K_Q \omega_z D} \quad (29)$$

In these equations, N_b is the number of blades, (F_x, F_y, F_z) is the force vector per unit area, ρ is the fluid density, N is the rotation rate in revolutions per seconds, ω_z is the rotation rate in radians per second and W_z is the free stream velocity. The flow in the free stream is parallel to the z-direction. The integration is carried out over the surface of the blade and the hub (which is also included in the force measurements). The abscissa in the following figures is the advance ratio:

$$J = \frac{W_z}{ND} \quad (30)$$

The design points of the propeller is at $J = 0.699$ (4021). Some differences occur at the extreme off-design conditions. The differences might be a result of insufficient grid resolution, shortcomings in the turbulence model for separated flows, or experimental deficiencies. Especially at low flow speed (small J), there is an influence of the propeller-induced flow on the inflow speed in the closed water tunnel.

Fig. 7.21 also includes sample computations based on a fine grid with 2107596 nodes. There is little influence of the grid density on the predicted performance characteristics of the propeller. This does not necessarily mean that the flow field is fully resolved on the coarser grid, but only that the major flow features are correctly captured.

Figs. 7.22 to 7.25 compare the computed and measured axial velocity profiles at 0.17 D behind the propeller 4021, where $Z=0$ at the generation line of the propeller. Numerical results are shown for the standard grid (250,000 nodes) and the fine grid (2,107,596 nodes). For all four radial locations, the overall level and the general shape of the velocity profiles agree well with the experimental data. Differences in the details of the wake are most likely due to deficiencies in the turbulence model or insufficient resolution. The differences in the computed

results on the two grids indicate that grid independence was not achieved on the standard grid. Another potential problem is the application of wall function boundary conditions at the blade surface. Wall functions do not allow for a consistent grid refinement in the near wall region, which could be the cause for the differences in the wake.

Best Practice Guidelines for Marine
 Applications of Computational Fluid Dynamics

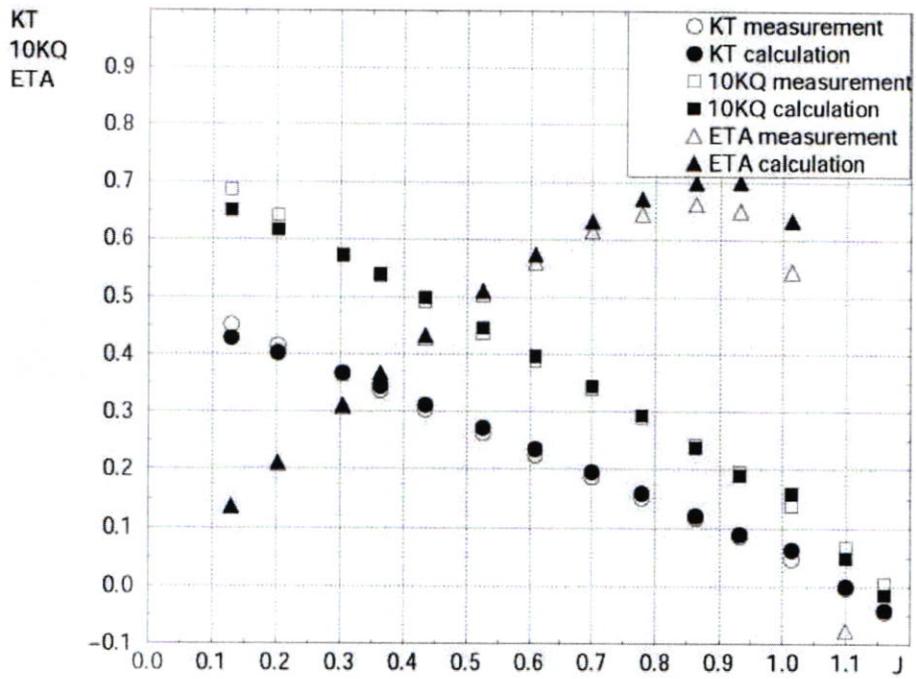


Figure 7.21: Comparison of K_T and K_Q for Propeller 4021

Best Practice Guidelines for marine Applications of Computational Fluid Dynamics

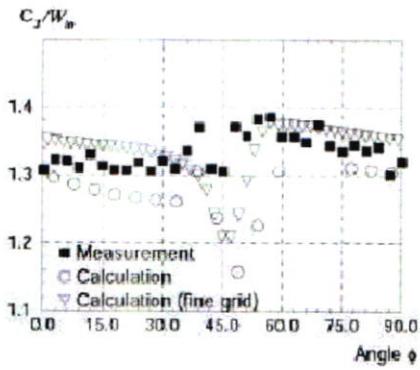


Figure 7.22 Radius 0.4: Axial velocity profiles at 0.17D behind the propeller 4021.

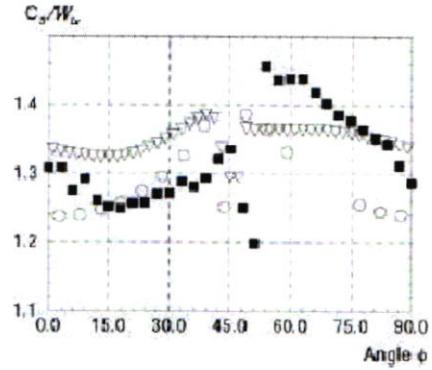


Figure 7.23 Radius 0.8: Axial velocity profiles at 0.17D behind the propeller 4021.

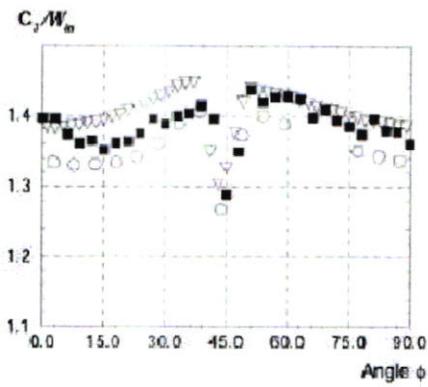


Figure 7.24 Radius 0.6: Axial velocity profiles at 0.17D behind the propeller 4021.

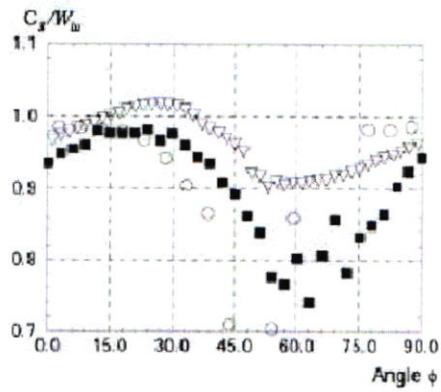


Figure 7.25 Radius 1.0: Axial velocity profiles at 0.17D behind the propeller 4021.

7.4.6. References

- ABDEL-MAKSOU, M.; BSCHORER, S.; SCHEUERER, G., (1995), "Numerische Berechnung der viskosen Strömung um einen rotierenden Propeller", Jahrbuch der Schiffbautechnischen Gesellschaft, Vol. 89, pp 332-339.
- ABDEL-MAKSOU, M.; MENTER, F.; SCHEUERER, G., (1996), "Numerische und experimentelle Untersuchung der viskosen Strömung um einen Skew-Propeller", Hauptversammlung der Schiffbautechnischen Gesellschaft, Bremen, 1996.
- ASC, (1995), "TASCflow User Documentation, Version 2.", Advanced Scientific Computing Ltd., Waterloo, Ontario, Canada, 1995.
- BLAUROCK, J.; LAMMERS, G., (1988), "The Influence of Profile Skew on the Velocity Field and Tip Vortex Shape in the Slipstream of Propellers", SNAME Propellers '88 Symposium.
- CHEN B., (1996), "Computational Fluid Dynamics of Four-Quadrant Marine-Propulsor Flow", M.Sc. thesis, Graduate College, University of Iowa.
- HEINKE, H.-J.; BOHM, M.; SCHMIDT, D., (1993), "Propeller aus Kohlefaserverbundstoffen", SVA Bericht 2010, Potsdam.
- KIM, T. H.; STERN, F., (1990), "Viscous Flow Around a Propeller-Shaft Configuration with Infinite-Pitch Rectangular Blades", Propulsion and Power, Vol. 6, No. 4, pp. 434-444.
- KROLL, N., (1989), "Berechnung von Strömungsfeldern um Propeller und Rotoren im Schwebeflug durch die Lösung der Euler-Gleichungen", DLR-FB 89-37, Braunschweig.
- LAUNDER, B. E.; SHARMA, B. I., (1974), "Application of the Energy Dissipation Model of Turbulence to the Calculation of the Flow Near a Spinning Disk", Letters in Heat and Mass Transfer, Vol. 1, No. 2.
- OH, K.-J.; KANG, S.-H., (1992), "Numerical Calculation of the Viscous Flow Around Rotating Marine Propeller", 19th ONR Symposium on Naval Hydrodynamics.
- PARK, W.-G.; SANKAR, N., (1993), "Numerical Simulation of the Incompressible Viscous Flow Around a Propeller", AIAA Paper 93-3503-CP.
- SÀNCHEZ-CAJA, A., (1996), "Numerical Calculation of Viscous Flow around DTRC Propeller 4119 for Advance Number Range 03.-1.1 Using the FINFLO Navier-Stokes Solver", Technical Report VALB141A, VTT Manufacturing Technology, Espoo, Finland.

8. Checklist of best practice advice for marine CFD

This section contains checklists of the best practice guidelines outlined in the previous chapters. A list of general guidelines relevant to generic CFD calculations is presented, followed by lists of guidelines specific to RANS calculations and potential flow calculations. The lists are presented such as to follow roughly the chronological sequence required to complete a CFD project.

8.1. General CFD guidelines

8.1.1. Guidelines on the training of CFD users

- A CFD user for non-routine applications should have good training and knowledge in classical fluid mechanics, a broad understanding of numerical methods, and detailed knowledge of the application being examined. This means that they will be able to understand the limitations of the particular models used (e.g. turbulence, boundary conditions, definition of Green's function being used).
- The training and education requirement for more routine applications can be less stringent, provided that clear guidelines or procedures have been established for the use of the code being used. An example of a routine application would be the simulation of a steady ship flow where many previous designs have been calculated and only relatively small changes in geometries and boundaries conditions occur.
- In both routine and non-routine applications, training on the use of the specific CFD code with the solution of realistic exercises is needed.

8.1.2. Guidelines on problem definition

- The user needs to give careful thought to the requirements and objectives of the simulation and typically might consider the following points:
 - Is a CFD simulation method really appropriate?
 - Are the objectives of the simulation clearly defined?
 - What are the requirements on accuracy?
 - What local/global quantities are needed from the simulation?
 - What are the documentation/reporting requirements?
 - What are the important flow physics involved?
 - What is the area of primary interest (domain) for the flow calculation?
 - Is the geometry well defined?
 - What level of validation is necessary? Is this a routine application, where validation and calibration has already been carried out on similar flow fields, and where only relatively small changes can be expected from earlier similar simulations? Or is it a non-routine application, where little earlier validation work has been done.
 - What level of computational resources is needed for the simulation (memory, disk space, CPU time) and are these available?
- Avoid over-simplification of the model which may omit important effects
- Be aware of the magnitude and implication of errors (e.g. round-off errors)
- Scale factor is important – solution is much easier at model scale (smaller Reynolds number) but there may be difficulties scaling up the results, i.e. Froude / Reynolds scaling differences.

8.1.3. Guidelines on global solution algorithm

- Check the adequacy of the solution procedure with respect to the physical properties of the flow.

- As a first step in this process, the parameters controlling convergence (e.g. relaxation parameters or Courant number) of the solution algorithm should be used as suggested by the CFD-code vendor or developer.
- If it is necessary to change parameters to aid convergence, it is not advisable to change too many parameters in one step, as it then becomes difficult to analyse which of the changes have influenced the convergence. In case of persistent divergence see sections on boundary conditions (section 3.7), grid (section 3.4), discretisation and convergence errors (section 3.2).
- Consider carefully whether the flow can be expected to exhibit steady or unsteady flow behaviour. Consider the size of the unsteady scales to be expected present in the flow field in comparison to the geometrical dimensions, and if this is large then an unsteady simulation is necessary.
- If a steady solution has been computed and there is a reason to be unsure that the flow is really steady, then an unsteady simulation should be carried out with the existing steady flow field as the initial condition. Examination of the time-development of the physical quantities in the locations of interest will identify whether the flow is steady or not.

8.1.4. Guidelines on the solution of the discretised equations

8.1.4.1. Guidelines on round off errors

- Always use the 64-bit representation of real numbers (double precision on common UNIX workstations).
- Developers are recommended to use the 64-bit representation of real numbers (REAL*8 in FORTRAN) as the default settings for their CFD code.

8.1.4.2. Guidelines on spatial discretisation

- Try to give an approximation of the numerical error in the simulation by applying a mesh or panel refinement study, or if this is not possible by mesh coarsening.
- If available in the code, make use of the calculation of an error estimator (which may be based on residuals, or on the difference between two solutions of different order accuracy).

8.1.4.3. Guidelines on temporal discretisation

- The overall solution accuracy is determined by the lower order component of the discretisation. At least second order accuracy is recommended in space and time. For time dependent flows the time and space discretisation errors are strongly coupled. Hence finer grids or higher order schemes are required (in both space and time).
- Check the influence of the order of the temporal discretisation by analysis of the frequency and time-development of a quantity of interest (e.g. the velocity in the main flow direction).
- Check the influence of the time-step on the results.
- Ensure that the time-step is adapted to the choice of the grid and the requested temporal size by resolving the frequency of the realistic flow and ensure that it complies with eventual stability requirements.

8.1.5. Guidelines on assessment of errors

- A potential source of user errors is in implementing the solution strategy with a particular code. Such errors might be minimised by the availability of a formal check list or by letting another CFD analyst checking through the code input data. The types of questions which should be considered are:

- Have the boundary conditions not only been properly defined, but also properly applied?
- Has the appropriate system of units been used?
- Is the geometry correct?
- Are the correct physical properties specified?
- Have the intended physical and mathematical models been used (e.g. gravity forces, rotation, user defined functions)?
- Have default parameters been changed which may affect the solution?
- Has the appropriate convergence criterion been defined and used?

8.1.6. Guidelines on analysis and interpretation of results

8.1.6.1. Guidelines on checking results

- Don't be seduced into believing that the solution is correct just because it has converged and produced high-quality colour plots (or even seductive video presentations) of the CFD simulations. Make sure that an elementary interpretation of the flow-field explains the fluid behaviour and that the trends of the flow analysis can be reconciled with a simple view of the flow.
- Check conserved variables, including an overall force/momentum balance.
- Check that velocities, forces, pressures, etc. have believable values.
- Check whether fluid variables such as velocity and pressure are smoothly distributed over the body and vary rapidly only where expected. Discontinuities may be the result of poor panel definition or insufficient mesh.
- Perform some simple hand calculations to check orders of magnitudes of variables.
- Run simple versions of the problem (e.g. with reduced geometry) to get an idea for the numbers involved.
- Make sure that the mean values of engineering parameters derived from the simulation are computed consistently (e.g. mass-average values, area-average values, time-average values). Calculation of local and mean engineering parameters with external post-processing software may be inconsistent with the solution method of the code used. Check that any test data used for comparison with the simulations is also computed in the same way as the data from the simulation.

8.1.6.2. Guidelines on the relevance of the results

- Consider whether the interpretation of the results and any decisions made, is within the accuracy of your computation.
- Ensure that the solution algorithm used is the most suitable, and recognise the approximations used.
- The accuracy of the solution will only be as good as the accuracy of the input conditions.
- Compare the result with similar problems, or simplified versions of the same problem.

8.1.6.3. Guidelines on further sensitivity studies

- Perform the calculation using several different panel and grid densities.
- Investigate the sensitivity of boundary conditions.
- If time permits run the problem using a different source code and compare the results.
- Investigate the effects of different viscous approximations or turbulence models.

8.1.7. Guidelines on documentation

- Keep good records of the simulation with clear documentation of assumptions, approximations, simplifications, geometry and data sources.

- Organise the documentation of the calculations so that another CFD expert can follow what has been done.
- Be aware that the level of documentation required depends strongly on the customers requirements as defined in the problem definition.

8.1.8. Guidelines on communication with code developer

8.1.8.1. Guidelines for the code developer and vendor

- A CFD user for non-routine applications should have good training and knowledge in classical fluid mechanics, a broad understanding of numerical methods, and detailed knowledge of the application being examined. This means that they will be able to understand the limitations of the models used (e.g. viscous effects, boundary conditions).
- The training and education requirement for more routine applications can be less stringent, provided that clear guidelines or procedures have been established for the use of the code being used. An example of a routine application would be the simulation of a standard component in a design environment where many previous designs have been calculated and only relatively small changes in geometries and boundaries conditions occur.
- In both routine and non-routine applications, training on the use of the specific CFD code with the solution of realistic exercises is needed.

8.1.8.2. Guidelines for the code user

- The user should recognise that codes can only be validated and verified for a class of problems involving specific variables. If the user is moving into an area where the code is not fully verified there is more risk of code errors.
- A suite of test cases set up and run by the user on new code releases provides an independent check on the code and highlights changes between releases (for example in default parameters).
- When a code error is suspected, the user should communicate this to the code vendor or developer as soon as possible, especially if no list of known bugs has been published. Other users may then profit from this experience or the user may find that the bug is well-known and a solution or work-around is available.
- In communication with the code developer or code vendor about a suspected program error, the user should provide a short concise description of the problem and all the necessary input data files so that the error can be reproduced. In cases where commercial sensitivity precludes this, special arrangements will need to be made.

8.2. RANS calculations

8.2.1. Guidelines on solution strategy

- Having established a clear problem definition, the user needs to translate this into a solution strategy involving issues and questions that have been addressed in the earlier chapters of this document, such as:
 - Mathematical and physical models.
 - Pressure or density based solution method.
 - Turbulence model.
 - Available code/solver.
 - Computational mesh.
 - Boundary conditions.

8.2.2. Guidelines on turbulence modelling

- The user should be aware that there is no universally valid general model of turbulence that is accurate for all classes of flows. Validation and calibration of the turbulence model is necessary for all applications.
- If possible, the user should examine the effect and sensitivity of results to the turbulence model by changing the turbulence model being used.
- The relevance of turbulence modelling only becomes significant in CFD simulations when other sources of error, in particular the numerical and convergence errors, have been removed or properly controlled. Clearly no proper evaluation of the merits of different turbulence models can be made unless the discretisation error of the numerical algorithm is known, and grid sensitivity studies become crucial for all turbulence model computations.

8.2.2.1. Guidelines on wall functions

- The meshing should be arranged so that the values of y^+ at all the wall adjacent mesh points is greater than 30 (the form usually assumed for the wall functions is not valid much below this value). It is advisable that the y^+ values do not exceed 100 and should certainly never be less than 11. Some commercial CFD codes account for this by switching to alternative functions if y^+ is < 30 . Be aware of this and check the user manuals.
- Cell centred schemes have their integration points at different locations in a mesh cell than cell vertex schemes. Thus the y^+ value associated with a wall adjacent cell differs according to which scheme is being used on the mesh. Care should be exercised when calculating the flow using different schemes or codes with wall functions on the same mesh.
- The values of y^+ at the wall adjacent cells strongly influence the prediction of friction and hence drag. Thus particular care should be given to the placement of near-wall meshing if these are important elements of the solution.
- Check that the correct form of the wall function is being used to take into account the wall roughness.

8.2.2.2. Guidelines on near wall resolution

- Make sure that the turbulence model being used is capable of resolving the flow structure through to the wall.
- The value of y^+ at the first node adjacent to the wall should be close to unity.
- Employ a small stretching factor for progressing the mesh spacing away from the wall. There should be at least ten mesh points between the wall and y^+ equal to 20.

8.2.2.3. Guidelines on weaknesses of the standard k- ϵ model

- The turbulent kinetic energy is over-predicted in regions of flow impingement and re-attachment leading to poor prediction of the development of flow around leading edges and bluff bodies. Kato and Launder [1993] have proposed a modification to the transport equation for ϵ which is designed to tackle this problem.
- Regions of re-circulation in a swirling flow are under-estimated. Reynolds Stress models (RSM) should be used to overcome this problem.
- Highly swirling flows are generally poorly predicted due to the complex strain fields. Reynolds Stress models (RSM) or non-linear eddy viscosity models should be used in these cases.
- Mixing is poorly predicted in flows with strong buoyancy effects or high streamline curvature. Reynolds Stress models should be used in these cases.
- Flow separation from surfaces under the action of adverse pressure gradients is poorly predicted. The real flow is likely to be much closer to separation (or more separated) than the calculations suggest. The Baldwin-Lomax one-equation model is often better than the

standard k- ϵ model in this respect, Baldwin and Lomax [1978]. The SST version of Menter's k- ω based, near wall resolved model mentioned in section 4.2.4 (Menter [1993, 1996]) also offers a considerable improvement.

- Flow recovery following re-attachment is poorly predicted. Avoid the use of wall functions in these regions.
- The spreading rates of wakes and round jets are predicted incorrectly. The use of non-linear k- ϵ models should be investigated for these problems.
- Turbulence driven secondary flows in straight ducts of non-circular cross section are not predicted at all. Linear eddy viscosity models cannot capture this feature. Use RSM or non-linear eddy viscosity modelling.
- Laminar and transitional regions of flow cannot be modelled with the standard k- ϵ model. This is an active area of research in turbulence modelling. No simple practical advice can be given other than advocating user intervention to switch the turbulence model on or off at predetermined locations.

8.2.3. Guidelines on definition of geometry

- Check and document that the geometry of the object being calculated is the geometry as intended. For example, the transfer of geometrical data from a CAD system to a CFD system may involve loss of surface representation accuracy. Visual display of the geometry helps here.
- In general, it is not necessary to explicitly include geometrical features that have dimensions below that of the local grid size provided that they are taken into account in the modelling (e.g. roughness in wall layer).
- In areas where local detail is needed then grid refinement in local areas with fine details should be used, such as in the neighbourhood of fine edges, or small clearance gaps. If grid refinement is used the additional grid points should lie on the original geometry and not simply be a linear interpolation of more grid points on the coarse grid.
- Check that the geometry is defined in the correct co-ordinate system and with the correct units which are requested by the CFD-code. CAD-systems often define the geometry in millimetres and this must be converted to SI-units if the code assumes that the geometry information is in these units. This is commonly done by most codes.
- If the geometry is altered or deformed by the hydrodynamic, mechanical or thermal loading, then some structural/mechanical calculation may be necessary to determine the exact geometry.

8.2.4. Guidelines on grids and grid design

- Clean up CAD geometry and for body fitted grids check that the surface grid conforms to the CAD geometry.
- When using periodic boundary conditions ensure high precision of the interface.
- Avoid highly skewed cells, in particular for hexahedral cells or prisms the included angles between the grid lines should be optimised in such a way that the angles are approximately 90 degrees. Angles with less than 40 or more than 140 degrees often show a deterioration in the results or lead to numerical instabilities, especially in the case of transient simulations.
- The angle between the grid lines and the boundary of the computational domain (the wall or the inlet- and outlet-boundaries) should be close to 90 degrees. This requirement is stronger than the requirement for the angles in the flow field far away from the domain boundaries.
- Avoid the use of tetrahedral elements in boundary layers.
- Away from boundaries, ensure that the aspect ratio (the ratio of the sides of the elements) is not too large. This aspect ratio should be typically not larger than 20. Near walls this restriction may be relaxed and indeed can be beneficial.

- The code requirements of mesh stretching or expansion ratios (rates of change of cell size for adjacent cells) should be observed. The change in mesh spacing should be continuous and mesh size discontinuities be avoided, particularly in regions of high gradients.
- The mesh should be finer in critical regions with high flow gradients, such as regions with high shear, and where there are significant changes in geometry or where suggested by error estimators. Make use of local refinement of the mesh in these regions, in accordance with the selected turbulence wall modelling (see Section 5.3). The location of a refinement interface should be away from high flow gradients.
- Check the assumption of regions of high flow gradients assumed for the grid with the result of the computation and rearrange grid points if found to be necessary.
- Analyse the suitability of the mesh by a grid dependency study (this could be local) where you use at least three different grid resolutions. If this is not feasible try to compare different order of spatial discretisations on the same mesh (see Section 3.5). The ITTC guidelines provide more detail in this area.
- Use the global topology of the mesh to help satisfy the above guidelines.

8.2.5. Guidelines on boundary conditions

8.2.5.1. General guidelines on boundary conditions

- Ensure that appropriate boundary conditions are available for the case being considered. For swirling flows consult manual to ensure appropriate boundary condition used (for example, radial equilibrium of pressure field instead of constant static pressure). Special non-reflecting boundary conditions are sometimes required for outflow and inflow boundaries where there are strong pressure gradients Giles [1990].
- Check whether the CFD code allows inflow at open boundary conditions. If inflow cannot be avoided at an open boundary then ensure that the transported properties of the incoming fluid including turbulence boundary conditions are properly modelled.
- Examine the possibilities of moving the domain boundaries to a position where the boundary conditions are more readily identified, are well-posed and can be precisely specified.
- For each class of problem an uncertainty analysis should be carried out in which the boundary conditions are systematically changed within certain limits to see the variation in results. Should any of these variations prove to have a sensitive effect on the simulated results and lead to large changes in the simulation, then it is clearly necessary to obtain more accurate data on the boundary conditions that are specified.

8.2.5.2. Guidelines on inlet conditions

- Examine the possibilities of moving the domain inlet boundaries to a position where the boundary conditions are easily identified, are well-posed and can be precisely specified.
- For each class of problem a sensitivity analysis should be carried out in which the inlet boundary conditions are systematically changed within certain limits. Aspects that should be examined are:
 - Inlet flow direction and magnitude.
 - Uniform inlet velocity (slug flow) or velocity profile.
 - Variation of physical parameters.
 - Variation of turbulence properties at inlet (see below).

8.2.5.3. Guidelines on specification of turbulence quantities at an inlet

- A particularly important issue is the specification of the turbulence properties at the inlet to the computational domain and verified quantities should be used as inlet boundary

conditions for turbulent kinetic energy k and dissipation ϵ , if these are available as the magnitude can significantly influence the results.

- If there are no data available, then the values need to be specified using sensible engineering assumptions, and the influence of the choice should be examined by sensitivity tests with different simulations.
- For the specification of the turbulent kinetic energy k , values should be used which are appropriate to the application. These values are generally specified through a turbulence intensity level. ERCOFTAC guidelines suggest a variety of values depending on flow type. In hydrodynamics, low "inlet" turbulence levels are likely, but zero turbulence will bring about anomalies in turbulence modelling unless specialised approaches to laminar and transitional regions are adopted.
- The specification of the turbulent length scale, as an equivalent parameter for the dissipation ϵ , is more difficult. For external flows, a value determined from the assumption that the ratio of turbulent and molecular viscosity μ_T/μ is of the order of 10 is appropriate. For simulations in which the near-wall region is modelled, for example in two layer modelling of boundary layers, the length scale should be based on the distance to the wall and be consistent with the internal modelling in the code.
- If more sophisticated distributions of k and ϵ are used these need to be consistent with the velocity profile, so that the production and dissipation term in the turbulence equations are in balance. An inconsistent formulation such as a constant velocity profile and constant profile of turbulence intensity at the inlet lead to an immediate unrealistic reduction of the turbulence quantities after the inlet. These can be checked by making a plot of the ratio of turbulent to molecular viscosity μ_T/μ . In cases where problems arise the inflow boundary should be moved sufficiently far from the region of interest so that an inlet boundary layer can develop.
- For RSM models the stresses themselves need to be specified, and as these are normally not available an assumption of isotropic flow conditions with zero shear stresses is generally made.

8.2.5.4. Guidelines on outlet conditions

- The boundary conditions imposed at the outlet should be selected to have a weak influence on the upstream flow. Extreme care is needed when specifying flow velocities and directions on the outlet plane. The most suitable outflow conditions are weak formulations involving specification of static pressure at the outlet plane.
- Particular care should be taken in strongly swirling flows where the pressure distribution on the outlet boundary is strongly influenced by the swirl, and cannot be specified independently of the swirl coming from upstream.
- Be aware of the possibility of inlet flow inadvertently occurring at the outflow boundary, which may lead to difficulties in obtaining a stable solution or even to an incorrect solution. If it is not possible to avoid this by relocating the position of the outlet boundary in the domain, then one possibility to avoid this problem is to restrict the flow area at the outlet, provided that the outflow boundary is not near the region of interest.
- If there are multiple outlets, then either pressure boundary conditions or mass flow specifications can be imposed depending on the known quantities.

8.2.5.5. Guidelines on solid walls

- Care should be taken that the boundary conditions imposed on solid walls are consistent with both the physical and numerical models used.
- If roughness on the wall is not negligible, significant levels of uncertainty can arise through incorrect specification of roughness within the wall function and when no detailed information is available great care is needed. Research in this area in ship hydrodynamics has been considerable.

8.2.5.6. Guidelines on symmetry and periodicity planes

- Symmetry and periodicity planes assume that the gradients perpendicular to the plane are either zero (for symmetry) or determined from the flow field (periodicity). If symmetry or periodicity planes cross the inlet or outlet boundaries then care should be taken to specify inlet or outlet variables that are consistent with these.

8.2.5.7. Guidelines on uncertainties with steady flow, symmetry and periodicity

- Check carefully whether the geometry is symmetric or whether a geometrical distortion or disturbance in the inlet conditions is present which can trigger asymmetric solutions.
- Estimate the Reynolds-number of the inflow and check whether the flow could be asymmetric, turbulent and/or unsteady (e.g. by sources or literature).
- After obtaining a steady solution, switch to the transient mode and check whether the solution remains stable.
- If there are difficulties to get a converged steady solution – especially if there is an oscillation of the residuals – switch to the transient mode.
- In case of doubt, the simulation should be unsteady and without symmetry assumptions as boundary conditions.

8.2.6. Guidelines on convergence

- Be aware that different codes have different definitions of residuals.
- Always check the convergence on global balances (conservation of mass, momentum and turbulent kinetic energy) where possible, such as the mass flow balance at inlet and outlet and at intermediate planes within the flow domain.
- Check not only the residual itself but also the rate of change of the residual with increasing iteration count.
- Convergence of a simulation should not be assessed purely in terms of the achievement of a particular level of residual error. Carefully define solution sensitive target quantities for the integrated global parameters of interest and select an acceptable level of convergence based on the rate of change of these (such as mass flow, lift, drag, and moment forces on a body).
- For each class of problem carry out a test of the effect of converging to different levels of residual on the integrated parameter of interest (this can be a single calculation that is stopped and restarted at different residual levels). This test demonstrates at what level of residual the parameter of interest can be considered to have converged and identifies the level of residual that should be aimed at in similar simulations of this class of problem.
- Monitor the solution in at least one point in a sensitive area to see if the region has reached convergence.
- For calculations that are proving difficult to converge, then the following advice may be helpful:
 - Use more robust numerical schemes during the first (transient) period of convergence and switch to more accurate numerical schemes as the convergence improves.
 - Reduce parameters controlling convergence, for instance under relaxation parameters or the CFL number.
 - If the solution is heavily under-relaxed increase relaxation factors at the end to see if the solution holds.
 - Check whether switching from a steady to a time-accurate calculation has any effect.
 - Consider using a different initial condition for the calculation.
 - Check the numerical and physical suitability of boundary conditions (see also Section 3.7.3 and Chapter 5)

- Check whether the grid quality in areas with large residual has any effect on the convergence rate.
- Look at the residual distribution and associated flow field for possible hints, e.g. regions with large residuals or unrealistic velocity levels.

8.3. Potential flow

8.3.1. Guidelines on definition of non-linear problems

- Linearised potential flow methods have limitations with regard to wave slope.
- Careful panel distribution is required at the vessel/free surface interface to provide enough resolution to resolve the wave profile sufficiently.
- Should wave breaking be possible within the solution, for example near the bow or at high speed, solutions may be unstable and require local grid coarsening to achieve a converged result.
- Control of the free surface panel size in the far field should take account of the effect of growing panel size on wave propagation and speed.

8.3.2. Guidelines on integration of viscous effects

- The use of empirical formula to estimate additional viscous effects should be used as an approximate method only, and care should be exercised in the choice of skin friction correlation line.
- Such methods can only be applied where the flow remains attached.
- For accurate resolution of stern wave and transom effects, where viscous forces are significant, empirical viscous approximations may not be sufficient.

8.3.3. Guidelines on definition of geometry

- Check that the geometry is defined in the correct co-ordinate system and with the correct units which are requested by the CFD-code. CAD-systems often define the geometry in millimetres and this must be converted to SI-units if the code assumes that the geometry information is in these units. This is commonly done by most codes.
- If the geometry is altered or deformed by the hydrodynamic or mechanical loading, then some structural/mechanical calculation may be necessary to determine the exact geometry.
- Ensure that panels edges meet exactly and that the body is totally enclosed, especially if importing body geometry from a CAD model.
- Grid refinement is required in areas of rapid pressure change.
- Flow separation will only occur wherever the user sets it to (i.e. where a wake sheet is applied).
- Careful panel definition is required at regions of high curvature (e.g. at the leading edge of propeller blades, fin stabilisers) to represent the body accurately. A finer distribution of panels should be used in regions likely to experience high fluid flow.
- The trailing edge must be located at a panel intersection to satisfy the Kutta condition. When defining panels around a section it may be easiest to start from the trailing edge.
- If the panels or the fluid domain are to be translated or rotated careful thought should be given to the location of the panels.
- If a cubic spline formulation is used care needs must be taken with the curve end conditions when trying to model sharp changes in direction.
- Adjacent bodies must not intersect or overlap.
- Panels should have a low aspect ratio and should not be highly skewed. Element sizes should vary gradually over the body. Should quadrilateral panels exhibit high levels of

- skew, they should be replaced by two triangular panels, blended to the surrounding panel size.
- Plate element normals must point outwards from the body.
 - Try to use the symmetry properties of the body geometry to the full.
 - For free surface flows at least 16 panels per wavelength are required for adequate resolution of the wave profile, and users should in any case perform mesh sensitivity studies to gain confidence in the results.
 - The wake sheet should extend far enough downstream to capture sufficient detail of the flow.
 - For propellers, the optimum chord-wise panel distribution will depend on the shape and radius of the leading edge.

8.3.4. Guidelines on boundary conditions

- Check that appropriate boundary conditions are available for the flow being modelled
- Ensure that waves are not reflected from the domain boundaries in time domain simulations.
- Systematic variation of boundary conditions e.g. the location of a radiation boundary, should be carried out to determine the uncertainty effects. If these effects are significant a more detailed analysis of the boundary conditions will be necessary.
- The wall boundary conditions will inherently be "free-slip" for a potential flow. If this is unsuitable, a different method or different viscous approximation should be used.

9. References

- AIAA (1988), "AIAA guide for the verification and validation of computational fluid dynamics simulations", AIAA G-077-1998.
- Aspley, D., Chen, W-L., Leschziner, M. & Lien, F-S. (1997), "Non-linear eddy-viscosity modelling of separated flows", IAHR J. Hydraulic Research, vol. 35, pp. 723-748.
- Baldwin, W.S. & Lomax, H., (1978), "Thin-layer approximation and algebraic model for separated turbulent flows", AIAA Paper 87-257.
- Bradshaw, P. (1994), "Turbulence: the chief outstanding difficulty of our subject", Experiments in Fluids, vol. 16, pp. 203-216.
- Cebeci, T. & Smith, A.M.O., (1974), "Analysis of turbulent boundary layers", Series in Appl. Math. & Mech., Vol. XV, Academic Press.
- Dawson, C.W. (1977), "A practical computer method for solving ship-wave problems", Proc. Of the 2nd Int. Conf. On Numerical Ship Hydrodynamics, Berkley, California.
- ERCOfTAC Best Practice Guidelines.
- Ferziger, J.H. & Peric, M. (1997), "Computational methods for fluid dynamics", Springer, Berlin.
- Fisher, E.M. & Rhodes, N. (1996), "Uncertainty in computational fluid dynamics", Proc. Mech. Eng., vol. 210, Part C: Journal of Mech. Eng. Sci., pp. 91-94.
- Fletcher, C.A.J. (1991), "Computational techniques for fluid dynamics, vol. I & II", Springer, Berlin.
- Giles, M.B., (1990), "Non-reflecting boundary conditions for Euler equation calculations", AIAA Journal, vol. 28, no. 12, pp. 2050-2058.
- Hess, J.L. & Smith, A.M.O. (1964), "Calculation of non-lifting potential flow around arbitrary three-dimensional bodies", Journal of Ship Research, vol. 8, no. 2.
- Hirsch, C. (1991), "Numerical computation of internal and external flows, vol. I & II", Wiley, New York.
- Kato, M. & Launder, B.E. (1993), "Three-dimensional modelling and heat-loss effects on turbulent flow in a nominally two-dimensional cavity", Int. J. Heat and Fluid Flow, vol. 16, pp. 171-177.
- Launder, B.E. & Spalding, D.B. (1972), "Mathematical models of turbulence", Academic Press.
- Launder, B.E. & Spalding, D.B. (1974), "The numerical computation of turbulent flow", Comp. Meth. In Appl. Mech. And Engng., vol. 3, pp. 269-289.
- Launder, B.E. (1984), "Second-moment closure: methodology and practice", Chapter in Turbulence Models and their Applications, vol. 2, Collection de la Direction des Etudes et Recherches d'Electricite de France, Eyrolles, Paris.
- Menter, F.R. (1993), "Zonal two equation k- ω turbulence models for aerodynamic flows", AIAA Paper 93-2906.
- Menter, F.R. (1994a), "Two-equation eddy-viscosity turbulence models for engineering applications", AIAA Journal, vol. 32, No. 8, pp. 1598-1605.
- Menter, F.R. (1994b), "Eddy viscosity transport equations and their relation to the k- ϵ model", NASA-TM-108854.
- Menter, F.R. (1996), "A comparison of some recent eddy-viscosity turbulence models", Transactions of the SNAME, vol. 118, pp. 514-519

Musker, A.J. (1988), "A panel method for predicting ship wave resistance", 17th Symp. Of Naval Hydrodynamics, Den Haag.

Patankar, S.V. (1980), "Numerical heat transfer and fluid flow", McGraw-Hill, New York.

Patel, V.C., Rodi, W. & Scheuerer, G. (1985), "Turbulence models for near-wall and low Reynolds number flows: a review", AIAA Journal, vol. 23, no. 9, pp. 1308-1318.

Reports to the 22nd ITTC.

Rizzi, A. & Voss, J. (1998), "Towards establishing credibility in computational fluid dynamics simulations", AIAA Journal, vol. 36, no. 5, pp. 668-675.

Roache, P.J. (1998), "Verification and validation in computational science and engineering", Hermosa Publishers, Albuquerque.

Rodi, W. (1981), "Progress in turbulence modelling for incompressible flows", AIAA Paper 81-45, St Louis.

Spalart, P.R. & Allmaras, S.R. (1992), "A one-equation turbulence model for aerodynamic flows", AIAA Paper, 92-0439.

Speziale, C.G. (1987a), "Second-order closure models for rotating and turbulent flows", Q. Appl. Math., vol. 45, pp. 69-71.

Tennekes, H. & Lumley, J.L. (1972), "A first course in turbulence", MIT Press, Cambridge. Latest edition 1994.

Wilcox, D.C. (1998), "Turbulence modelling for CFD", DCW Industries, Inc.

Wolfstein, M.W. (1969), "The velocity and temperature distribution in a one-dimensional flow with turbulence augmentation and pressure gradient", Int. J. Heat and Mass Transfer, vol. 12, pp. 301-312.