Incompressible computational fluid dynamics

Edited by

MAX D. GUNZBURGER
and
ROY A. NICOLAIDES

Incompressible Computational Fluid Dynamics
Trends and Advances

Edited by

MAX D. GUNZBURGER
Virginia Polytechnic Institute and State University

and

ROY A. NICOLAIDES
Carnegie Mellon University

CAMBRIDGE
UNIVERSITY PRESS

Published by the Press Syndicate of the University of Cambridge
The Pitt Building, Trumpington Street, Cambridge CB2 1RP
40 West 20th Street, New York, NY 10011-4211, USA
10 Stamford Road, Oakleigh, Melbourne 3166, Australia

First published 1993

Printed in the United States of America

Library of Congress Cataloging-in-Publication data is available.

A catalog record for this book is available from the British Library

ISBN 0-521-40407-X hardback

Preface

Numerical methods for incompressible fluid dynamics have developed to the point at which a survey of the field is both timely and appropriate. A major stimulus to the field has been the large number of applications in which incompressible flows play a crucial role, and this has spurred the interest of numerous computational engineers and mathematicians. The articles which follow provide a reasonably broad view of algorithmic and theoretical aspects of incompressible flow calculations.

It should be noted at the outset that it can be dangerous to define an algorithm for simulating incompressible flows by setting, for example, the density to be constant in a successful compressible flow algorithm. The nature of the pressure as a Lagrange multiplier rather than as a thermodynamic variable as well as the infinite speed of propagation of disturbances and other factors peculiar to incompressible flows, make algorithmic development and implementation in this context a unique undertaking (see Appendix 7A).

Perhaps the first major advance in the application of large scale digital computation to incompressible flows occured in the late 1950s with the introduction of staggered mesh techniques, exemplified, for example, by the Marker-and-Cell (MAC) scheme. The use of staggered meshes in the context of the primitive variable formulation was found to provide a stable discretization of the incompressibility constraint. Shortly thereafter, it was realized that the use of staggered meshes could be avoided by employing the streamfunction-vorticity formulation in which the incompressibility constraint does not explicitly appear. Numerous finite difference algorithms were proposed and used based on this formulation of the Navier-Stokes equations.

Despite the sucess of the stream function-vorticity approach, many difficulties associated with practical computations remained unresolved, including complex geometries, boundary condition treatments and flows in three dimensions. The success of finite element methods in addressing these problems in the structural mechanics context naturally led to interest in applying them to incompressible flows. Thus, starting in the early 1970s and continuing to the present day, there has been an evergrowing understanding and use of finite element methods for incompressible flow problems.

A different methodology, whose modern development started during the 1960s, involves the use of point vortex and related singular functions to approximate solutions of the Navier-Stokes equations at relatively small viscosities. These methods have the

apparent advantage of being grid-free, at least for certain types of problems involving simple boundaries. Vortex methods are currently enjoying a period of intense development.

More recently, the potentially high accuracy possible with spectral methods has generated considerable interest in their application to incompressible flows. The need to account for complicated boundaries naturally leads to the spectral element method which in turn is closely related to the p-version of the finite element method.

Another recent development has been covolume algorithms employing Voronoi-Delaunay dual tesselations of general domains. The discrete equations of the covolume technique can be chosen so that when specialized to rectangular geometries, they coincide with those of classical staggered mesh methods. In this way a generalization of the MAC scheme to triangular and tetrahedral domains is obtained.

The chapters in the book are arranged alphabetically. The chapter by Engelman provides an overview of some of the many real world applications in which the numerical simulation of incompressible flows plays a significant role.

A relatively large proportion of the papers is devoted to specific algorithms, or to components of algorithms for incompressible flows. Thus, Puckett reviews the state-of-the-art in vortex methods while Karniadakis, Orszag, Rønquist, and Patera do likewise for spectral methods. The latter also give a brief account of lattice gas methods. Nicolaides describes the use of Voronoi-Delaunay tesselations as a basis for covolume discretizations of the equations of incompressible flow. Temam considers methods based on notions stemming from dynamical systems theory.

There are four papers addressing basic algorithmic issues in finite element methods for incompressible flows. Dean and Glowinski give a review of some of the large body of theoretical results concerning the finite element discretization and solution of incompressible flows. Thatcher focuses on algorithms and elements that have proved to be useful for three-dimensional flows; Verfürth reviews some recent developments in adative mesh-refinement techniques based on a posteriori error estimation; Franca, Hughes, and Stenberg give a thorough account of "stabilized" finite element methods for which some difficulties associated with discretizing the incompressibility constraint are circumvented.

Cardot, Mohamadi, and Pironneau discuss how to efficiently incorporate classical turbulence models into existing laminar flow codes. Gunzburger discusses the application of finite element analyses and methodologies to optimal design and control problems for incompressible flows.

There are three chapters devoted to implementation issues in the context of engineering applications. Löhner discusses practical aspects of the design of incompressible flow solvers; Hafez and Soliman discuss issues arising in both the discretization and solution

phases of an unstaggered grid method; a similar treatment of an algorithm using pressure dissipation is given by Habashi, Peeters, Robichaud, and Nguyen.

There are some aspects of incompressible computational fluid dynamics that are not considered, or are only briefly considered in the book. A non-exhaustive list of these includes the application of wavelets, cellular automata, turbulence modeling and details about the direct simulation of turbulent flows. We apologize to anyone offended by these omissions; in some cases we tried to obtain contributions but were thwarted by unwilling or tardy authors; in other cases we felt that the subject matter has not reached a sufficient stage of development to merit inclusion in this volume. We also note incompressible computational fluid dynamics is still a developing field so that necessarily, due to the time it takes to put together such a book, some very recent developments could not be included.

We, the editors, wish to take this opportunity to thank all the authors who contributed to this book; we feel that we have gathered here a group of papers by outstanding authorities and which are representative of the current state of computational methods for incompressible flows. We also wish to thank our editor at Cambridge University Press, Lauren Cowles, for her patience, understanding, and encouragement over the duration of the project.

Acknowledgement

We would like to express our sincere thanks to the United States Air Force Office of Scientific Research (M. D. G. and R. A. N.) and the Office of Naval Research (M. D. G.) for supporting our research in incompressible CFD over the past several years.

M. D. Gunzburger, Blacksburg

R. A. Nicolaides, Pittsburgh

Contributing Authors

B. Cardot
STCAN
Paris, France

Edward J. Dean
Department of Mathematics, University of Houston
Houston, TX 77204, USA

Roland Glowinski
Department of Mathematics, University of Houston
Houston, TX 77204, USA

Max D. Gunzburger
Department of Mathematics, Virginia Polytechnic Institute and State University
Blacksburg, VA 24061, USA

Michael S. Engelman
Fluid Dynamics International
Evanston, IL 60201, USA

Leopoldo P. Franca
Laboratório Nacional de Computação Científica
22290 Rio de Janeiro, Brazil

W. G. Habashi
Concordia University
Montreal, Quebec, Canada

M. Hafez
Department of Mechanical and Aerospace Engineering
University of California at Davis
Davis, CA 95616, USA

L. Steven Hou
Department of Mathematics and Statistics, York University
Toronto, Ontario M3J 1P3, Canada

Thomas J. R. Hughes
Division of Applied Mechanics, Stanford University
Stanford, CA 94305, USA

George Em Karniadakis
Department of Mechanical and Aerospace Engineering, Princeton University
Princeton, NJ 08544, USA

Rainald Löhner
School of Engineering and Applied Science, The George Washington University
Washington, DC 20052, USA

B. Mohammadi
INRIA
78153 Le Chesnay, France

V-N. Nguyen
Computational Methods Group, Pratt & Whitney
Montreal, Canada

Roy A. Nicolaides
Department of Mathematics, Carnegie Mellon University
Pittsburgh, PA 15213, USA

Steven A. Orszag
Department of Mechanical and Aerospace Engineering, Princeton University
Princeton, NJ 08544, USA

Anthony T. Patera
Department of Mechanical Engineering, Massachusetts Institute of Technology
Cambridge, MA 02139, USA

M. F. Peeters
Computational Methods Group, Pratt & Whitney
Montreal, Canada

Oliver Pironneau
INRIA
78153 Le Chesnay, France

Elbridge Gerry Puckett
Department of Mathematics, University of California at Davis
Davis, CA 95616, USA

M. P. Robichaud
Computational Methods Group, Pratt & Whitney
Montreal, Canada

Einar M. Rønquist
Nektonics, Inc.
Cambridge, MA, USA

M. Soliman
Department of Mechanical and Aerospace Engineering
University of California at Davis
Davis, CA 95616, USA

Rolf Stenberg
Faculty of Mechanical Engineering, Helsinki University of Technology
02150 Espoo, Finland

Thomas P. Svobodny
Department of Mathematics and Statistics, Wright State University
Dayton, OH 45435, USA

Roger Temam
Department of Mathematics, Indiana University
Bloomington, IN 47405, USA

R. W. Thatcher
Department of Mathematics, UMIST
Manchester M60 1QD, UK

R. Verfürth
Institut für Angewandte Mathematik, Universität Zürich
Zürich CH-8001, Switzerland

Contents

1 A Few Tools For Turbulence Models In Navier-Stokes Equations

B. Cardot, B. Mohammadi, and O. Pironneau

Abstract

This article is for those who have already a computer program for incompressible viscous transient flows and want to put a turbulence model into it. We discuss some of the implementation problems that can be encountered when the Finite Element Method is used on classical turbulence models except Reynolds stress tensor models. Particular attention is given to boundary conditions and to the stability of algorithms.

1.1 Introduction

Many scientists or engineers turn to turbulence modeling after having written a Navier-Stokes solver for laminar flows.

For them turbulence modeling is an external module into the computer program. Generally, the main ingredients to built a good Navier-Stokes solver are known; this includes tools like mixed approximations for the velocity u and pressure p to avoid checker board oscillations and also upwinding to damp high Reynolds number oscillations; however the problems that one may meet while implementing a turbulence model are not so well known because these models have not been studied much theoretically.

Judging from the literature [3][11][12][15][19][22] the most commonly used turbulence models seem to be

- algebraic eddy viscosity models (zero equation models)
- $k-\varepsilon$ models (two equations models)
- Reynolds stress models

All three start from a decomposition of u and p into a mean part and a fluctuating part u'. However oscillations are understood either as time oscillations or space oscillations or even variations due to changes in initial conditions. In any case, the decomposition $u+u'$ is applied to the Navier-Stokes equations. After averaging (and several handwaving steps) some closed set of equations are obtained for the mean flow variables (still denoted u and p here). We do not intend to discuss the validity of the models or how well they compares with experiments; we want to discuss the discretization of the equations and the

stability properties if any. So we shall take them one by one and make a few comments along the way. Note however that this paper is by no means a review as the literature is way too rich on this subject.

In Section 1.2 we shall start with the easiest, the algebraic models of Smagorinsky [21] and Baldwin-Lowmax [2]. Wall laws will also be discussed in this section.

Then in Section 1.3, we shall discuss the $k-\varepsilon$ model and its variations. Emphasis will be on the positivity of the variables.

In Section 1.4 some numerical results are presented including a $k-\varepsilon$ simulation of the flow behing a cylinder at moderate Reynolds number.

We shall not discuss the Reynolds stress models because they are still controversial and because it appears that they may be easier to implement in a compressible flow solver directly. This is because they make the Navier-Stokes equations hyperbolic even in the case of incompressible fluids. Also the complexity of the equations of the $R_{ij}-\varepsilon$ model makes the mathematical analysis quite difficult and messy.

For laminar flow the Navier-Stokes equations are

$$\partial_t u + u\nabla u + \nabla p - \mu\Delta u = 0 \tag{1.1.1}$$

$$\nabla \cdot u = 0 \text{ in } \Omega\times]0,T[\tag{1.1.2}$$

$$u = u_\Gamma \text{ on } \Gamma\times]0,T[(\Gamma = \partial\Omega) \tag{1.1.3}$$

$$u|_{t=0} = u^o \text{ in } \Omega \tag{1.1.4}$$

In our numerical tests these have been discretized by piecewise biquadratic quadrilateral elements for u and piecewise linear discontinuous triangular elements for the pressure p (each quadrilateral is divided into two triangles) (see [9][18] or [23] for example).

Furthermore upwinding was implemented by using the characteristic Galerkin method [5], [8], [17].

Then (1.1.1)–(1.1.4) is approximated by

$$\frac{1}{\delta t}(u_h^{m+1}, v_h) - (p_h^{m+1}, \nabla\cdot v_h) + \nu(\nabla u_h^{m+1} : \nabla v_h) = \frac{1}{\delta t}(u_h^m o X_h^m, v_h), \quad \forall v_h \in V_{oh} \tag{1.1.5}$$

$$(\nabla \cdot u_h^{m+1}, q_h) = 0, \quad \forall q_h \in Q_h \tag{1.1.6}$$

Here δt is the time step size, (f,g) stands for $\int_\Omega f(x)g(x)dx$, V_{oh} is the space of piecewise biquadratic velocities on the quadrangulation of Ω which are zero on Γ; Q_h is the space of piecewise linear discontinuous pressure on the triangulation of Ω. Finally, $X_h^m(x)$ is an approximation at t^n of the solution of

$$\frac{dX}{d\tau} = u_h^m(X_h,\tau); \quad X_t(t^{m+1}) = x; \quad X_h^m(x) \simeq X(t^m) \tag{1.1.7}$$

Note that $X_h^m(x) \simeq x - u_h^m(x)\delta t$ and that $(u_h^{m+1}(x) - u_h^m o X_h^m)/\delta t$ is an approximation of $\partial_t u_h + u_h \nabla u_h$.

Alternatively we could have used a Galerkin least square upwinding or a Newton method without upwinding with an implicit in time discretization of $\partial_t u$. Other popular elements for spatial discretizations include the $P^1 - iso - P^2/P^1$ element or the mini-element [1] on triangles. Much of what will be said applies also if these alternative choices are made.

One advantage of (1.1.5)–(1.1.6) is that it yields a symmetric linear system at each time step; the price to pay is the computation of a complicated integral: $\int_\Omega u_h^n(X_h^n(x))v_h(x)dx$.

The linear system has the form

$$\begin{pmatrix} A & B \\ B^T & 0 \end{pmatrix}\begin{pmatrix} U \\ P \end{pmatrix} = \begin{pmatrix} W \\ 0 \end{pmatrix} \qquad A = \begin{pmatrix} D & 0 \\ 0 & D \end{pmatrix} \quad B = \begin{pmatrix} B^1 \\ B^2 \end{pmatrix} \tag{1.1.8}$$

where v is the vector of degrees of freedom of u_h and P for p_h. The matrix A is also a bloc matrix where

$$D_{ij} = \frac{1}{\delta_t}(w^i, w^j) + \nu(\nabla w^i, \nabla w^j) \tag{1.1.9}$$

where w^i is the hat finite element scalar basis function at vertex i. Thus the two components of u_h are coupled through the pressure only by $B_{ij}^k = -(q^i, \partial w^j/\partial x_k)$.

1.2 Zero Equation Models

1.2.1 Eddy viscosity

Most zero equation models consist of the Navier-Stokes equations

$$\partial_t u + u\nabla u + \nabla p - \nabla \cdot [\mu_T(\nabla u + \nabla u^T)] = 0 \tag{1.2.1}$$

$$\nabla \cdot u = 0 \tag{1.2.2}$$

with a non constant viscosity (eddy viscosity) μ_T which is a given function of position x and velocity μ.

In his subgrid scaled model Smagorinsky [21] (see also [7][15][20]) suggested

$$\mu_T = \nu + ch^2|\nabla u + \nabla u^T| \tag{1.2.3}$$

where ν is the molecular (reduced) viscosity c is a numerical constant ($c = 0.01$) and $h(x)$ is the average mesh size around x. In order to have a reasonably smooth function $h(x)$, one method is to define it at the vertices of the quadrangulation as the average of the edges which are issued from the vertex.

The Baldwin-Lomax [2] model is well suited to turbulent boundary layers. There ν_T is also an algebraic function of $\nabla \times u$ and the distance $y(x)$ between x and Γ.

$$\nu_T = l^2|\nabla \times u| \text{ if } y \leq y_c, \ = \frac{0.0269\delta_i}{(1+5.5(\frac{y}{\delta}))^6} \text{ otherwise} \tag{1.2.4}$$

If $\kappa = 0.41$ is the Von-Karman constant, $l = \kappa y(1 - e^{-\frac{y}{A}})$ with $A = 26(\nu/\frac{\partial u}{\partial n})^{1/2}$. The boundary layer thickness is δ. Finally $\delta_i = \int_0^\delta (1 - \frac{u}{u(\delta)})dy$ and y_c is such that both functions in (1.2.4) match.

To discretize (1.2.1), (1.2.2) it is usually sufficient to take ν_T at time m in the equations which define u^{m+1}. Thus the analogue of (1.1.5), (1.1.6) is

$$\begin{aligned} &\frac{1}{\delta t}(u_h^{m+1}, v_h) - (p_h^{m+1}\nabla \cdot v_h) \\ &\quad + \tfrac{1}{2}(\nu_T^m[\nabla u_h^{m+1} + (\nabla u_h^{m+1})^T] : [\nabla v_h + \nabla v_h^T]) = \tfrac{1}{\delta t}(u_h^m o X_h^m, v_h) \end{aligned} \tag{1.2.5}$$

$$(\nabla \cdot u_h^{m+1}, q_h) = 0. \tag{1.2.6}$$

There are two additional difficulties here

- the matrix of the linear system now depends upon m through μ_T^m
- the components of u_h^{m+1} are coupled through the viscous terms also.

These difficulties may be removed by considering the scheme

$$\begin{aligned} &\tfrac{1}{\delta t}(u_h^{m+1}, v_h) - (p_h^{m+1}\nabla \cdot v_h) + (\bar{\nu}_T \nabla u_h^{m+1} : \nabla v_h) = \\ &\tfrac{1}{\delta t}(u_h^m o X_h^m, v_h) + (\bar{\nu}_T \nabla u_h^{m+1} : \nabla v_h) - \tfrac{1}{2}(\mu_h^m([\nabla u_h^m + \nabla u_h^{mT}] : [\nabla v_h + \nabla v_h^T]) \end{aligned} \tag{1.2.7}$$

where $\bar{\nu}_T$ is close to μ_h^m and recomputed every say 5 or 6th time step.

Convergence however is not guaranteed, unlike (1.2.3), (1.2.5) which is more stable [13] than (1.1.5), (1.1.6).

1.2.2 Wall laws

Equations (1.2.1), (1.2.2) may develop boundary layers near the walls. An attempt can be made to remove them from the computational domain by replacing the no slip condition (1.1.3) by slip conditions of the type

$$u \cdot n = 0 \tag{1.2.8}$$

$$\nu_T \tfrac{\partial u}{\partial n} \cdot \tau + \alpha u \cdot \tau = \beta \tag{1.2.9}$$

where n is the outward normal to Γ, τ is the tangent, and α, β may be non linear function of $u \cdot \tau$ and even ∇u. Parès [16] has shown that such boundary conditions lead to a well

posed problem for the Navier-Stokes equations provided some conditions on the growth of α and β are satisfied.

Condition (1.2.8) must be enforced at vertices but n is defined on the edges of Γ; some sort of average is needed to define n at the vertices.

Another way is to notice that when $u \in H^1(\Omega)^2$

$$(u, \nabla q) = 0, \quad \forall q \in H^1(\Omega) \Rightarrow \nabla \cdot u = 0 \text{ in } \Omega \textbf{ and } u \cdot n|_\Gamma = 0 \tag{1.2.10}$$

because

$$(u, \nabla q) = -(\nabla \cdot u, q) + \int_\Gamma u \cdot nq\,. \tag{1.2.11}$$

So consider the implementation of (1.2.8)–(1.2.9) in weak form in (1.2.7) in which the normal n has completly disappeared:

$$\begin{aligned} \frac{1}{\delta t}(u_h^{m+1}, v_h) - (\nabla p_h^{m+1}, v_h) + (\bar{\nu}_T \nabla u_h^{m+1}, \nabla v_h) + \int_\Gamma \alpha^m u_h^{m+1} v_h = \\ f(v_h) + \int_\Gamma \beta^m v_h \cdot \tau \end{aligned} \tag{1.2.12}$$

for all v_h continuous piecewise bilinear

$$(u_h^{m+1}, \nabla q_h) = 0 \quad \forall q_h \text{ piecewise linear.} \tag{1.2.13}$$

In (1.2.13), $f(v_h)$ denotes the right hand side of (1.2.7). Numerical test with this formulation can be found in Parès [16] and in [4].

1.3 The $k - \varepsilon$ Model

Let k the turbulent kinetic energy and ε the turbulent rate of dissipated energy, so if u' denotes the time oscillations of u:

$$k = \frac{1}{2} < |u'|^2 > \tag{1.3.1}$$

$$\varepsilon = \frac{\nu}{2} < |\nabla u' + \nabla u'^T|^2 >; \tag{1.3.2}$$

In the $k - \varepsilon$ model it is assumed that the small time oscillations of u, p are equivalent to an eddy viscosity:

$$\mu_T = c_\mu \frac{k^2}{\varepsilon} \tag{1.3.3}$$

and that, away from the walls $k - \varepsilon$ are governed by:

$$k_{,t} + u\nabla k - \frac{c_\mu}{2}\frac{k^2}{\varepsilon}|\nabla u + \nabla u^T|^2 - \nabla \cdot (c_\mu \frac{k^2}{\varepsilon}\nabla k) + \varepsilon = 0 \tag{1.3.4}$$

$$\varepsilon_{,t} + u\nabla \varepsilon - \frac{c_1}{2} k|\nabla u + \nabla u^T|^2 - \nabla \cdot (c_\varepsilon \frac{k^2}{\varepsilon}\nabla \varepsilon) + c_2 \frac{\varepsilon^2}{k} = 0 \tag{1.3.5}$$

with $c_\mu = 0.09$, $c_1 = 0.1296$, $c_2 = 1.92$, $c_\varepsilon = 0.07$.

Natural boundary conditions could be

$$k, \varepsilon \text{ given at t = 0; } k|_\Gamma = k_\Gamma = 0, \quad \varepsilon|_\Gamma = \varepsilon_\Gamma \tag{1.3.6}$$

however ε_Γ is not known so the model is not well posed near the solid walls. A coupling with a one equation model (unknown ε) near the walls can be done. Alternatively an attempt is usually made to remove the low Reynolds regions from the computational domain by applying *wall laws*

$$k|_\Gamma = u^{*2} c_\mu^{-\frac{1}{2}}, \quad \varepsilon|_\Gamma = \frac{u^{*3}}{\kappa\delta} \tag{1.3.7}$$

$$u \cdot n = 0, \quad \alpha u \cdot \tau + \beta \frac{\partial u \cdot \tau}{\partial n} = \gamma \tag{1.3.8}$$

where κ is the Von Karman constant ($\kappa = 0.41$), δ the grid size at the wall (an approximation of the boundary layer thickness), u^* (computed by (1.3.9)) the friction velocity, $\beta = c_\mu k^2/\varepsilon$, $\alpha = \beta/[\kappa\delta(B + \kappa^{-1}\log(\delta/D))]$ where D is a roughness constant, $\gamma = -u^*|u^*|$ and B is such that (1.3.8) matches the viscous sublayer. To compute u*, Reichard's law may be used:

$$u^* = \frac{u.\tau}{f(u^*)}; \quad f(u^*) = 2.5log(1 + 0.4y^+) + 7.8(1 - e^{-\frac{y^+}{11}} - \frac{y^+}{11} e^{-0.33y^+}) \tag{1.3.9}$$

with $y^+ = \delta u^*/\nu$. So α, β, γ in (1.3.8) are nonlinear functions of $u \cdot \tau$. For smooth walls an easier alternative is $\alpha = \gamma = 0$.

For physical and mathematical reasons it is essential that the system (1.3.1)–(1.3.5) yields positive values for k and ε.

We shall now show that if the system has a smooth solution for given positive initial data and positive Dirichlet conditions on the boundaries then k and ε stay positive and bounded at later times. For this purpose one looks at

$$\theta = \frac{k}{\varepsilon}. \tag{1.3.10}$$

If D_t denotes the total derivative operator, $\partial/\partial t + u\nabla$ and E denotes $\frac{1}{2}|\nabla u + \nabla u^T|^2$, then

$$\begin{aligned} D_t\theta &= \frac{1}{\varepsilon} D_t k - \frac{k}{\varepsilon^2} D_t \varepsilon \\ &= \theta^2 E(c_\mu - c_1) - 1 + c_2 + c_\mu \nabla \cdot \frac{k^2}{\varepsilon} \nabla\theta + 2c_\mu \theta^2 \nabla\theta \cdot \nabla(\frac{k}{\theta}) \\ &\quad + (c_\mu - c_\varepsilon)\frac{k}{\varepsilon^2} \nabla \cdot \frac{k^2}{\varepsilon} \nabla\varepsilon \end{aligned} \tag{1.3.11}$$