

Cambridge University Press

978-1-107-01895-2 - Computational Fluid Dynamics for Engineers

Bengt Andersson, Ronnie Andersson, Love Håkansson, Mikael Mortensen, Rahman Sudiyo and Berend van Wachem

Frontmatter

[More information](#)

Computational Fluid Dynamics for Engineers

Computational fluid dynamics (CFD) has become an indispensable tool for many engineers. This book gives an introduction to CFD simulations of turbulence, mixing, reaction, combustion and multiphase flows. The emphasis on understanding the physics of these flows helps the engineer to select appropriate models with which to obtain reliable simulations. Besides presenting the equations involved, the basics and limitations of the models are explained and discussed. The book, combined with tutorials, project and Power-Point lecture notes (all available for download), forms a complete course. The reader is given hands-on experience of drawing, meshing and simulation. The tutorials cover flow and reactions inside a porous catalyst, combustion in turbulent non-premixed flow and multiphase simulation of evaporating sprays. The project deals with the design of an industrial-scale selective catalytic reduction process and allows the reader to explore various design improvements and apply best practice guidelines in the CFD simulations.

Bengt Andersson is a Professor in Chemical Engineering at Chalmers University, Sweden. His research has focused on experimental studies and modelling of mass and heat transfer in various chemical reactors ranging from automotive catalysis to three-phase flow in chemical reactors.

Ronnie Andersson is an Assistant Professor in Chemical Engineering at Chalmers University. He obtained his PhD at Chalmers in 2005 and from 2005 until 2010 he worked as a consultant at Epsilon HighTech as a specialist in CFD simulations of combustion and multiphase flows. His research projects involve physical modelling, fluid-dynamic simulations and experimental methods.

Love Håkansson works as a consultant at Engineering Data Resources – EDR in Oslo, Norway. His research has been in mass transfer in turbulent boundary layers. He is now working on simulations of single-phase and multiphase flows.

Mikael Mortensen obtained his PhD at Chalmers University in 2005 in turbulent mixing with chemical reactions. After two years as a post doc at the University of Sydney, he is now working with fluid dynamics at the Norwegian Defence Research Establishment in Lillehammer, Norway.

Rahman Sudiyo is a Lecturer at the University of Gadjah Mada in Yogyakarta, Indonesia. He received his PhD at Chalmers University in 2006. His research has been in multiphase flow.

Berend van Wachem is a Reader at Imperial College London in the UK. His research projects involve multiphase flow modelling, ranging from understanding the behaviour of turbulence on the scale of individual particles to the large-scale modelling of gas–solid and gas–liquid flows.

Cambridge University Press
978-1-107-01895-2 - Computational Fluid Dynamics for Engineers
Bengt Andersson, Ronnie Andersson, Love Håkansson, Mikael Mortensen, Rahman Sudiyo and Berend van Wachem
Frontmatter
[More information](#)

Cambridge University Press

978-1-107-01895-2 - Computational Fluid Dynamics for Engineers

Bengt Andersson, Ronnie Andersson, Love Håkansson, Mikael Mortensen, Rahman Sudiyo and Berend van Wachem
Frontmatter

[More information](#)

Computational Fluid Dynamics for Engineers

BENGT ANDERSSON

Chalmers University, Sweden

RONNIE ANDERSSON

Chalmers University, Sweden

LOVE HÅKANSSON

Engineering Data Resources – EDR, Norway

MIKAEL MORTENSEN

Norwegian Defence Research Establishment, Norway

RAHMAN SUDIYO

University of Gadjah Mada, Indonesia

BEREND VAN WACHEM

Imperial College London, UK



CAMBRIDGE
UNIVERSITY PRESS

Cambridge University Press
978-1-107-01895-2 - Computational Fluid Dynamics for Engineers
Bengt Andersson, Ronnie Andersson, Love Håkansson, Mikael Mortensen, Rahman Sudiyo and Berend van Wachem
Frontmatter
[More information](#)

CAMBRIDGE UNIVERSITY PRESS
Cambridge, New York, Melbourne, Madrid, Cape Town,
Singapore, São Paulo, Delhi, Tokyo, Mexico City

Cambridge University Press
The Edinburgh Building, Cambridge CB2 8RU, UK

Published in the United States of America by Cambridge University Press, New York

www.cambridge.org
Information on this title: www.cambridge.org/9781107018952

© B. Andersson, R. Andersson, L. Håkansson, M. Mortensen, R. Sudiyo, B. van Wachem, L. Hellström 2012

This publication is in copyright. Subject to statutory exception
and to the provisions of relevant collective licensing agreements,
no reproduction of any part may take place without the written
permission of Cambridge University Press.

First published 2012

Printed in the United Kingdom at the University Press, Cambridge

A catalogue record for this publication is available from the British Library

Library of Congress Cataloguing in Publication data
Computational fluid dynamics for engineers / Bengt Andersson . . . [et al].
p. cm.

Includes bibliographical references and index.
ISBN 978-1-107-01895-2 (hardback)
1. Fluid dynamics. 2. Engineering mathematics. I. Andersson, Bengt, 1947 June 15–
TA357.C58776 2011
532'.05 – dc23 2011037992

ISBN 978-1-107-01895-2 Hardback

Additional resources for this publication at
www.cambridge.org/9781107018952

Cambridge University Press has no responsibility for the persistence or
accuracy of URLs for external or third-party internet websites referred to
in this publication, and does not guarantee that any content on such
websites is, or will remain, accurate or appropriate.

Contents

	<i>Preface</i>	<i>page</i>
1	Introduction	1
	1.1 Modelling in engineering	1
	1.2 CFD simulations	1
	1.3 Applications in engineering	2
	1.4 Flow	2
	1.4.1 Laminar flow	3
	1.4.2 Turbulent flow	3
	1.4.3 Single-phase flow	4
	1.4.4 Multiphase flow	4
	1.5 CFD programs	4
2	Modelling	8
	2.1 Mass, heat and momentum balances	9
	2.1.1 Viscosity, diffusion and heat conduction	9
	2.2 The equation of continuity	12
	2.3 The equation of motion	14
	2.4 Energy transport	16
	2.4.1 The balance for kinetic energy	16
	2.4.2 The balance for thermal energy	18
	2.5 The balance for species	18
	2.6 Boundary conditions	18
	2.6.1 Inlet and outlet boundaries	19
	2.6.2 Wall boundaries	19
	2.6.3 Symmetry and axis boundary conditions	20
	2.6.4 Initial conditions	20
	2.6.5 Domain settings	21
	2.7 Physical properties	21
	2.7.1 The equation of state	22
	2.7.2 Viscosity	22

3	Numerical aspects of CFD	24
3.1	Introduction	24
3.2	Numerical methods for CFD	25
3.2.1	The finite-volume method	25
3.2.2	Geometrical definitions	26
3.3	Cell balancing	26
3.3.1	The convective term	27
3.3.2	The diffusion term	28
3.3.3	The source term	28
3.4	Example 1 – 1D mass diffusion in a flowing gas	29
3.4.1	Solution	29
3.4.2	Concluding remarks	33
3.5	The Gauss–Seidel algorithm	33
3.6	Example 2 – Gauss–Seidel	34
3.7	Measures of convergence	37
3.8	Discretization schemes	38
3.8.1	Example 3 – increased velocity	39
3.8.2	Boundedness and transportiveness	40
3.8.3	The upwind schemes	40
3.8.4	Taylor expansions	42
3.8.5	Accuracy	43
3.8.6	The hybrid scheme	44
3.8.7	The power-law scheme	45
3.8.8	The QUICK scheme	45
3.8.9	More advanced discretization schemes	46
3.9	Solving the velocity field	47
3.9.1	Under-relaxation	49
3.10	Multigrid	50
3.11	Unsteady flows	51
3.11.1	Example 4 – time-dependent simulation	52
3.11.2	Conclusions on the different time discretization methods	57
3.12	Meshing	58
3.12.1	Mesh generation	58
3.12.2	Adaptation	60
3.12.3	Numerical diffusion	60
3.13	Summary	61
4	Turbulent-flow modelling	62
4.1	The physics of fluid turbulence	62
4.1.1	Characteristic features of turbulent flows	63
4.1.2	Statistical methods	66
4.1.3	Flow stability	69
4.1.4	The Kolmogorov hypotheses	70

	Contents	vii
4.1.5	The energy cascade	72
4.1.6	Sources of turbulence	74
4.1.7	The turbulent energy spectrum	75
4.2	Turbulence modelling	76
4.2.1	Direct numerical simulation	79
4.2.2	Large-eddy simulation	79
4.2.3	Reynolds decomposition	81
4.2.4	Models based on the turbulent viscosity hypothesis	86
4.2.5	Reynolds stress models (RSMs)	96
4.2.6	Advanced turbulence modelling	99
4.2.7	Comparisons of various turbulence models	99
4.3	Near-wall modelling	99
4.3.1	Turbulent boundary layers	101
4.3.2	Wall functions	104
4.3.3	Improved near-wall-modelling	107
4.3.4	Comparison of three near-wall modelling approaches	109
4.4	Inlet and outlet boundary conditions	110
4.5	Summary	112
5	Turbulent mixing and chemical reactions	113
5.1	Introduction	114
5.2	Problem description	115
5.3	The nature of turbulent mixing	117
5.4	Mixing of a conserved scalar	119
5.4.1	Mixing timescales	119
5.4.2	Probability density functions	120
5.4.3	Modelling of turbulent mixing	124
5.5	Modelling of chemical reactions	130
5.5.1	$Da \ll 1$	130
5.5.2	$Da \gg 1$	131
5.5.3	$Da \approx 1$	138
5.6	Non-PDF models	141
5.7	Summary	142
6	Multiphase flow modelling	143
6.1	Introduction	144
6.1.1	Characterization of multiphase flows	144
6.1.2	Coupling between a continuous phase and a dispersed phase	146
6.2	Forces on dispersed particles	147
6.3	Computational models	149
6.3.1	Choosing a multiphase model	150
6.3.2	Direct numerical simulations	151
6.3.3	Lagrangian particle simulations, the point-particle approach	152

viii	Contents	
	6.3.4 Euler–Euler models	155
	6.3.5 The mixture model	156
	6.3.6 Models for stratified fluid–fluid flows	158
	6.3.7 Models for flows in porous media	160
6.4	Closure models	161
	6.4.1 Interphase drag	161
	6.4.2 Particle interactions	163
	6.4.3 Heat and mass transfer	168
6.5	Boundaries and boundary conditions	169
	6.5.1 Lagrangian dispersed phase	169
	6.5.2 Eulerian dispersed phase	170
6.6	Summary	171
	6.6.1 Guidelines for selecting a multiphase model	172
7	Best-practice guidelines	174
	7.1 Application uncertainty	175
	7.1.1 Geometry and grid design	175
	7.2 Numerical uncertainty	175
	7.2.1 Convergence	175
	7.2.2 Enhancing convergence	176
	7.2.3 Numerical errors	176
	7.2.4 Temporal discretization	177
	7.3 Turbulence modelling	177
	7.3.1 Boundary conditions	177
	7.4 Reactions	178
	7.5 Multiphase modelling	178
	7.6 Sensitivity analysis	180
	7.7 Verification, validation and calibration	180
	<i>Appendix</i>	181
	<i>References</i>	185
	<i>Index</i>	186

Cambridge University Press

978-1-107-01895-2 - Computational Fluid Dynamics for Engineers

Bengt Andersson, Ronnie Andersson, Love Håkansson, Mikael Mortensen, Rahman Sudiyo and Berend van Wachem
Frontmatter[More information](#)

Preface

Computational fluid dynamics (CFD) has become an indispensable tool for engineers. CFD simulations provide insight into the details of how products and processes work, and allow new products to be evaluated in the computer, even before prototypes have been built. It is also successfully used for problem shooting and optimization. The turnover time for a CFD analysis is continuously being reduced since computers are becoming ever more powerful and software uses ever more efficient algorithms. Low cost, satisfactory accuracy and short lead times allow CFD to compete with building physical prototypes, i.e. ‘virtual prototyping’.

There are many commercial programs available, which have become easy to use, and with many default settings, so that even an inexperienced user can obtain reliable results for simple problems. However, most applications require a deeper understanding of fluid dynamics, numerics and modelling. Since no models are universal, CFD engineers have to determine which models are most appropriate to the particular case. Furthermore, this deeper knowledge is required since it gives the skilled engineer the capability to judge the potential lack of accuracy in a CFD analysis. This is important since the analysis results are often used to make decisions about what prototypes and processes to build.

Our ambition is that this book will provide sufficient background for CFD engineers to solve more advanced problems involving advanced turbulence modelling, mixing, reaction/combustion and multiphase flows. This book presents the equations that are to be solved, *but, more importantly*, the essential physics in the models is described, and the limitations of the models are discussed. In our experience, the most difficult part for a CFD engineer is not to select the best numerical schemes but to understand the fluid dynamics and select the appropriate models. This approach makes the book useful as an introduction to CFD irrespective of the CFD code that is used, e.g. finite-volume, finite-element, lattice Boltzmann etc.

This book requires a prior knowledge of transport phenomena and some understanding of computer programming. The book (and the tutorials/project) is primarily intended for engineering students. The objective is to teach the students how to do CFD analysis correctly but not to write their own CFD code. Beyond this, the book will give an understanding of the strengths and weakness of CFD simulations. The book is also useful for experienced and practicing engineers who want to start using CFD themselves or, as project managers, purchase these services from consulting firms.

We have added several questions of reflective character throughout the book; it is recommended that you read these to confirm that the most important parts have

Cambridge University Press

978-1-107-01895-2 - Computational Fluid Dynamics for Engineers

Bengt Andersson, Ronnie Andersson, Love Håkansson, Mikael Mortensen, Rahman Sudiyo and Berend van Wachem
Frontmatter[More information](#)

been understood. However, the book intentionally contains few simulation results and worked-through examples. Instead we have developed three tutorials and one larger project that give students the required hands-on experience. The tutorials take 6–8 hours each to run; the project takes 30 hours to run and write a report. These tutorials and the project are available from the authors. We have chosen to use a commercial code (ANSYS/Fluent) in our course, but the problem formulations are written very generally and any commercial program could be used. Our experience is that commercial CFD programs can be obtained for teaching purposes at very low cost or even free. An alternative is to use an open-source program, e.g. OpenFoam. Unfortunately, the user interface in OpenFoam is not as well developed as are those in the commercial programs, so students will have additional problems in getting their programs working.

This book has successfully been used in a CFD course at Chalmers University since 2004. Every year approximately 60 chemical and mechanical engineering students take the course. Over the years this book has also been used for PhD courses and courses in the industry. The text has been rewritten every year to correct errors and in response to very valuable suggestions from the students. PowerPoint lecture notes are also available from the authors.

Scope

Chapter 1 provides an introduction to what can be solved with a CFD program and what inputs are required from the user. It also gives an insight into what kind of problems are easy or difficult to solve and how to obtain reliable results.

Chapter 2 contains the equations that are solved by the CFD software. The student should know these equations from their prior courses in transport phenomena, but we have included them because they are the basis for CFD and an up-to-date knowledge of them is essential.

In *Chapter 3* the most common numerical methods are presented and the importance of boundedness, stability, accuracy and convergence is discussed. We focus on the finite volumes on which most commercial software is based and only a short comparison with the finite-element method is included. There is no best method available for all simulations since the balance among stability, accuracy and speed depends on the specific task.

Chapter 4 gives a solid introduction to turbulence and turbulence modelling. Since simulation of turbulent flows is the most common application for engineers, we have set aside a large part of the chapter to describe the physics of turbulence. With this background it is easier to present turbulence modelling, e.g. why sources and sinks for turbulence are important. The k – ε model family, k – ω , Reynolds stress and large-eddy models are presented and boundary conditions are discussed in detailed.

Chapter 5 carefully analyses turbulent mixing, reaction and combustion. The physics of mixing is presented and the consequence of large fluctuations in concentration is discussed. A probability distribution method is presented and methods to solve instantaneous, fast and slow kinetics are formulated. A simple eddy-dissipation model is also presented.

In *Chapter 6* multiphase models are presented. First various tools with which to characterize multiphase flow and forces acting on particles are presented. Eulerian–Lagrangian, Euler–Euler, mixture (algebraic slip), volume-of-fluid and porous-bed models are presented and various closures for drag, viscosity etc. are formulated. Simple models for mass and heat transfer between the phases are also presented.

Finally, *Chapter 7* contains a best-practice guideline. It is based on the guidelines presented by the European Research Community on Flow Turbulence and Combustion, ERCOFTAC, in 2000 and 2009 for single-phase and multiphase systems, respectively.

In *Tutorial 1* reactions inside a spherical porous catalyst particle are studied. The reaction is exothermic and flow, heat and species must be modelled. The student will learn how to draw and mesh a two-dimensional (2D) geometry. They will also specify boundary conditions and select the models to solve. The kinetics is written as a user-defined function (UDF) and the student will learn how to implement a UDF in a CFD simulation. Convergence is a problem, and the student will learn about physical reaction instability, numerical instabilities, under-relaxation and numerical diffusion. In the report the student is required to show that their simulations fulfil the criteria given in the best practice guidelines.

In *Tutorial 2* turbulent mixing and combustion in a bluff-body stabilized non-premixed turbulent flame is studied. An instantaneous adiabatic equilibrium reaction, i.e. combustion of methane in air, is simulated. The student should select an appropriate turbulence model and solve for flow, turbulence, mean mixture fraction, mixture fraction variance, species and heat with appropriate boundary conditions, e.g. wall functions. Mesh adaptation to obtain the proper y^+ for the wall functions is introduced. The students should analyse whether jets and recirculations exist in the flow and whether the reaction is fast compared with turbulent mixing.

In *Tutorial 3* a spray is modelled using an Eulerian–Lagrangian multiphase model. Continuous phase and spray velocity combined with heat transfer and evaporation are modelled. The student should analyse the fluid–spray interaction and choose what forces should be included in the model.

The *Project* is dedicated to the design of an industrial-scale selective catalytic reduction (SCR) process. The student generates a three-dimensional (3D) computer-aided design (CAD) model and mesh, analyses the performance, and suggests and evaluates design improvements of the SCR reactor.

Acknowledgements

We are very grateful to the students who have given us very valuable feedback and thus helped us to improve the book. We would also like to thank Mrs Linda Hellström, who did the graphics, and Mr Justin Kamp, who corrected most of our mistakes in the English language.

Bengt Andersson