

# Preface

The majority of flows encountered in nature and engineering applications are turbulent. Although turbulence has been studied as part of classical mechanics for over a century, it is still one of the unsolved problems in physics and remains to be an active area of research. The field of study that uses numerical simulation to examine fluid dynamics is called Computational Fluids Dynamics (CFD). There is great demand in utilizing CFD for analyzing problems ranging from turbulent flows around aircraft and ground vehicles to larger scale problems related to weather forecasting and environmental assessment. Such demands are very likely to grow in the coming years as the engineering community at large pursues improvements in energy efficiency and performance for various fluid-based systems.

Presently, there are several commercial CFD solvers released with turbulence analysis capability. With software creating beautiful visualizations of turbulent flows, it may appear that any type of flows can be numerically predicted. While there may be some truth to such capability, it is still difficult to solve most turbulent flow problems without relying on companion experiments. That raises a question of why we still are not able to perfectly predict the behavior of turbulent flows. The dynamics of turbulent flows obeys the Navier–Stokes equations, upon which CFD solvers are based. However, turbulence exhibits flow structures over a wide range of spatial and temporal scales that all interact amongst them in a complex nonlinear manner. That means that the spatial grid must be fine enough to resolve the smallest scales in turbulent flows while ensuring that the computational domain is large enough to encompass the largest flow structures. Such grid requirement becomes increasingly costly as we tackle flows at a higher Reynolds number. Despite the significant improvement in the computational capability with recent high-performance computers, we still do not expect computers to be able to handle these large grids for very high Reynolds number flows.

For this particular reason, turbulence is not likely to be completely solved in the near future. Flow physics taking place at scales below the resolvable scales must be represented with appropriate models, referred to as turbulence models. Presently, there is not a universally accepted turbulence model or numerical algorithm that can yield a solution unaffected by discretizations of the flow field. Thus, CFD should

continue to be an active field of research with efforts focused towards predicting the essential features of turbulent flows with turbulence models. As such, engineers and scientists using CFD must understand how the governing equations are numerically solved. We must also be equipped with the ability to correctly interpret the numerical solution. With these points in mind, we should construct a necessary and sufficient computer program appropriate to simulate the fluid flow of interest. For commercial software, sufficient details on the solver technique should be provided in the reference manual so that users can determine whether the solver can be appropriately used for the problem at hand.

This book describes the fundamental numerical methods and approaches used to perform numerical simulations of turbulent flows. The materials presented herein are aimed to provide the basis to accurately analyze unsteady turbulent flows. This textbook is intended for upper level undergraduate and graduate students who are interested in learning CFD. This book can also serve as a reference when developing incompressible flow solvers for those already active in CFD research. It is assumed that readers have some knowledge of fluid mechanics and partial differential equations. This textbook does not assume the readers to have advanced knowledge of numerical analysis.

This textbook aims to enable readers to construct his or her own CFD code from scratch. The present textbook covers the numerical methods required for CFD and places emphasis on the incompressible flow solver with detailed discussions on discretization techniques, boundary conditions, and turbulent flow physics. The introduction to CFD and the governing equations are offered in Chap. 1, followed by the coverage of basic numerical methods in Chap. 2. Incompressible flow solvers are derived and discussed in detail in Chaps. 3 and 4. We also provide discussions on the immersed boundary methods in Chap. 5. A brief overview on turbulent flows is given in Chap. 6 with details needed for analyzing turbulent flows using Reynolds-Averaged Navier–Stokes equations (RANS) and Large-Eddy Simulation (LES) provided in Chaps. 7 and 8, respectively. At the end of the book, an appendix is attached to offer details on the generalized coordinate system, Fourier analysis, and modal decomposition methods.

A large portion of the present book is based on the material taught over the years by the first author for the course entitled “Computational Fluid Dynamics and Turbulent Flows” at Osaka University and his textbook entitled, “Numerical Simulations of Turbulent Flows” (1st and 2nd editions in Japanese) that has been available in Japan since 1999. Chapters 1–4 and 6–8 as well as Appendices A and B in the present book are founded heavily on the Japanese version by Kajishima. The present textbook enjoys additions of stability analysis (Sect. 2.5), immersed boundary methods (Chap. 5), and modal decomposition methods (Sect. 6.3.7 and Appendix C) by Taira based on the courses taught at the Florida State University. Furthermore, exercises have been added after each chapter to provide supplemental materials for the readers.

The preparation of this book has benefited greatly from comments, feedback, and encouragements from Takashi Ohta, Shintaro Takeuchi, Takeshi Omori, Yohei Morinishi, Shinnosuke Obi, Hiromochi Kobayashi, Tim Colonius, Clarence

Rowley, Steven Brunton, Shervin Bagheri, Toshiyuki Arima, and Yousuff Hussaini. The stimulating discussions with them on various topics of CFD over the years have been invaluable in putting together the materials herein. We must also thank our research group members and students for providing us with detailed comments on the drafts of this book that helped improve the organization and correctness of the text. Finally, we would like to acknowledge Michael Luby and Brian Halm at Springer for working with us patiently and Nobuyuki Miura and Kaoru Shimada at Yokendo for their support on the earlier Japanese versions.

Osaka, Japan  
Tallahassee, Florida, USA  
July 2016

Takeo Kajishima  
Kunihiko Taira

Computational Fluid Dynamics  
Incompressible Turbulent Flows

Kajishima, T.; Taira, K.

2017, XV, 358 p. 107 illus., Hardcover

ISBN: 978-3-319-45302-6