

1. Record Nr.	UNINA9910634038603321
Autore	Lecheler Stefan
Titolo	Computational fluid dynamics : getting started quickly with ANSYS CFX 18 through simple examples / / Stefan Lecheler
Pubbl/distr/stampa	Cham, Switzerland : , : Springer, , [2022] ©2022
ISBN	9783658384531 9783658384524
Descrizione fisica	1 online resource (211 pages)
Disciplina	620.1064
Soggetti	Computational fluid dynamics
Lingua di pubblicazione	Inglese
Formato	Materiale a stampa
Livello bibliografico	Monografia
Nota di bibliografia	Includes bibliographical references and index.
Nota di contenuto	Intro -- Preface -- Symbol directory -- Contents -- 1: Introduction -- 1.1 Aim of This Book -- 1.2 Tasks of the Numerical Flow Calculation -- 1.3 Structure of the Book -- 2: Conservation Equations of Fluid Mechanics -- 2.1 Aim of This Chapter -- 2.2 Derivation of the Conservation Equations -- 2.2.1 Equation of Conservation of Mass -- 2.2.2 Conservation of Momentum Equations -- 2.2.3 Conservation of Energy Equation -- 2.3 Navier-Stokes Equations -- 2.3.1 Complete Navier-Stokes Equations -- Navier-Stokes Equations in Scalar Form (Cartesian Coordinates) -- Navier-Stokes Equations in Vector Form (Cartesian Coordinates) -- Navier-Stokes Equations in Divergence Form -- 2.3.2 Additionally Required Equations and Quantities -- 2.3.3 The Substance Values -- Boundary Conditions -- Physical Boundary Conditions at the Inflow Edge -- Physical Boundary Conditions at the Downstream Edge -- Physical Boundary Conditions at the Solid State Boundary -- Non-Reflective Boundary Conditions -- Summary of Boundary Conditions -- 2.3.4 Reynolds-Averaged Navier-Stokes Equations -- 2.3.5 Turbulence Models -- 2.4 Simplification Possibilities -- 2.4.1 Introduction -- 2.4.2 Thin-Layer Navier-Stokes Equations -- 2.4.3 Euler Equations -- 2.4.4 Boundary Layer Equations -- 2.4.5 Potential Equation -- 3: Discretization of the Conservation Equations -- 3.1 Aim of This Chapter -- 3.2 What Does Discretization Mean? -- 3.3 Spatial Discretization -- 3.3.1 Discretization of the First Derivatives --

3.3.2 Discretization of the Second Derivatives -- 3.3.3 Notes on Spatial Discretization -- 3.4 Time Discretization -- 3.4.1 Time Asymptotic or Stationary Solutions -- 3.4.2 Time-Accurate or Transient Solutions -- 3.5 Difference Equations -- 3.5.1 Derivation -- 3.5.2 Consistency, Stability and Convergence -- Consistency -- Stability -- Convergence -- 3.5.3 Additive Numerical Viscosity. 3.5.4 Upwind Discretization -- 3.5.5 Explicit and Implicit Discretization -- 3.5.6 CFL Number -- 3.5.7 Summary -- 4: Computational Meshes -- 4.1 Aim of This Chapter -- 4.2 Overview -- 4.3 Structured Meshes -- 4.3.1 Cartesian Meshes -- 4.3.2 Curvilinear Meshes -- 4.3.3 The Transformation of Coordinates into Curvilinear Coordinates -- 4.3.4 Block Structured Meshes -- 4.4 Unstructured Meshes -- 4.5 Mesh Adaptation -- 4.5.1 Mesh Densification -- 4.5.2 Adaptive Meshes -- 5: Solution Methods -- 5.1 Aim of This Chapter -- 5.2 Overview -- 5.3 Central Methods -- 5.3.1 Overview -- 5.3.2 Lax-Wendroff Method -- 5.3.3 Runge-Kutta Multi-Step Method -- 5.3.4 ADI Method -- 5.4 Upwind Methods -- 5.4.1 Overview -- 5.4.2 Flux Vector Splitting Method -- 5.4.3 Flux Difference Splitting Method -- 5.4.4 Summary -- 5.5 High-Resolution Methods -- 5.5.1 Overview -- 5.5.2 Monotonicity, TVD and Entropy Condition -- 5.5.3 Limiter Functions -- 5.5.4 Summary -- 5.6 Comparison of the Methods -- 5.6.1 Stationary Flow Through a Divergent Nozzle -- 5.6.2 Unsteady Flow in a Shock Wave Tube -- 6: Typical Workflow of a Numerical Flow Calculation -- 6.1 Aim of This Chapter -- 6.2 Overview -- 6.3 Generation of the Calculation Area (Geometry) -- 6.4 Generation of the Mesh (Meshing) -- 6.5 Preparation of the Flow Calculation (Setup) -- 6.6 Flow Calculation (Solution) -- 6.7 Evaluation (Results) -- 6.8 Validation -- 6.9 Introduction to the Exercise Examples -- 6.10 The ANSYS WORKBENCH Working Environment -- 7: Example Airfoil Flow -- 7.1 Generation of the Calculation Area (Geometry) -- 7.2 Generation of the Mesh (Meshing) -- 7.2.1 Starting the MESHING Program and Creating a Standard Mesh -- 7.2.2 Refinement of the Computational Mesh on the Profile -- 7.2.3 Associative Naming of the Boundaries -- 7.2.4 Exiting the MESHING Program -- 7.3 Preparation of the Flow Calculation (Setup). 7.3.1 Starting the CFX-PRE Program -- 7.3.2 Definition of the Calculation Parameters -- 7.3.3 Definition of the Physical Boundary Conditions -- 7.3.4 Inflow Boundary -- 7.3.5 Outflow Boundary -- 7.3.6 Solid Boundary -- 7.3.7 Symmetry Planes -- 7.3.8 Periodic Boundary Conditions -- 7.3.9 Exiting the CFX-PRE Program -- 7.4 Calculation of the Flow (Solution) -- 7.4.1 Starting the CFX-SOLVER Program -- 7.4.2 Monitoring Convergence Behaviour -- 7.4.3 Exiting the CFX-SOLVER Program -- 7.5 Evaluation (Results) -- 7.5.1 Starting the CFD-POST Program -- 7.5.2 Generation of Isoline Images -- 7.5.3 Vector Image Creation -- 7.5.4 Streamline Image Generation -- 7.5.5 Generation of Diagrams -- 7.5.6 Calculation of Integral Values -- 7.5.7 Preparation of a Report -- 7.5.8 Exiting the CFD-POST Program -- 8: Example Internal Pipe Flow -- 8.1 Generation of the Calculation Area (Geometry) -- 8.1.1 Importing a CAD File -- 8.1.2 Generation of the Computational Domain with the Program DESIGN MODELER -- 8.2 Generation of the Mesh (Meshing) -- 8.2.1 Starting the MESHING Program and Creating a Standard Mesh -- 8.2.2 Refinement of the Mesh on the Pipe Walls -- 8.2.3 Associative Naming of the Boundaries -- 8.2.4 Exiting the MESHING Program -- 8.3 Preparation of the Flow Calculation (Setup) -- 8.3.1 Starting the CFX-PRE Program -- 8.3.2 Definition of the Calculation Parameters -- 8.3.3 Definition of the Physical Boundary Conditions -- 8.3.4 Inflow Rim in Front -- 8.3.5 Inflow Rim Top -- 8.3.6 Outflow Boundary -- 8.3.7 Solid Boundary --

8.3.8 Symmetry Plane -- 8.3.9 Exiting the CFX-PRE Program -- 8.4 Calculation of the Flow (Solution) -- 8.4.1 Starting the CFX-SOLVER Program -- 8.4.2 Monitoring Convergence Behaviour -- 8.4.3 Exiting the CFX-SOLVER Program -- 8.5 Evaluation (Results) -- 8.5.1 Starting the CFD-POST Program -- 8.5.2 Generation of Isoline Images. 8.5.3 Vector Image Creation -- 8.5.4 Streamline Image Generation -- 8.5.5 Exiting the CFD-POST Program -- 9: Example Double Tube Heat Exchanger -- 9.1 Generation of the Calculation Area (Geometry) -- 9.2 Generation of the Mesh (Meshing) -- 9.2.1 Starting the MESHING Program and Creating a Standard Mesh -- 9.2.2 Refinement of the Mesh on the Pipe Walls -- 9.2.3 Associative Naming of the Border Areas -- 9.2.4 Exiting the MESHING Program -- 9.3 Preparation of the Flow Calculation (Setup) -- 9.3.1 Starting the CFX-PRE Program -- 9.3.2 Definition of the Calculation Parameters -- 9.3.3 Definition of the Physical Boundary Conditions -- 9.3.4 Inflow Boundaries for Fluid Inside and Fluid Outside -- 9.3.5 Outlet Boundaries for Fluid Inside and Fluid Outside -- 9.3.6 Solid Boundaries for Fluid Outer and Tube End Faces -- 9.3.7 Symmetry Planes for Fluid Inside, Fluid Outside and Pipe -- 9.3.8 Interface Planes Between Fluid and Pipe -- 9.3.9 Exiting the CFX-PRE Program -- 9.4 Calculation of the Flow (Solution) -- 9.4.1 Starting the CFX-Solver Program -- 9.4.2 Monitoring Convergence Behaviour -- 9.4.3 Exiting the CFX-Solver Program -- 9.5 Evaluation (Results) -- 9.5.1 Starting the CFD-POST Program -- 9.5.2 Generation of Isoline Images -- 9.5.3 Vector Image Creation -- 9.5.4 Generation of Diagrams -- 10: Example Parameter Variation -- Answers to the Target Control -- Answers to Sect. 2.1 Conservation Equations of Fluid Mechanics -- Answers to Sect. 3.1 Discretization of the Conservation Equations -- Answers to Sect. 4.1 Computational Meshes -- Answers to Sect. 5.1 Solution Methods -- Answers to Sect. 6.1 Typical Workflow of a Numerical Flow Calculation -- References -- Index.
