Outline for CFD Text Book through Spoken Tutorials

The chapters in the book will follow this outline.

1. Chapter 1: Installing and running OpenFOAM and ParaView-(6-7 Pages)

1.1 Introduction to OpenFOAM-(0.5 Page)

1.2 Introduction to Paraview-(0.5 Page)

1.3 Install OpenFOAM and Paraview through Synaptic Package Manager-(1-2 Pages)

1.4 Install OpenFOAM and Paraview through openfoam website-(1 Page)

1.5 Configure the installed software-(0.5 Page)

1.6 Test the installed software-(0.5 Page)

1.7 Creating user of working directory(0.5-1 Page)

1.8 Running OpenFOAM and Paraview with an example of Lid Driven Cavity-This will explain the problem case file-(1 Page)

2. Chapter 2: Creating simple geometry in OpenFOAM-(5-6 Pages)

2.1 Simple geometry-Description of the simple geometry and how it can be blocked-(1 Page)

2.2 BlockMeshDict file in OpenFOAM-(1 Page)

2.3 Co-ordinate numbering of the geometry-This explains how the geometry co-ordinates should be numbered to form blocks and faces-(0.5 Page)

2.4 Meshing parameters-This explains the parameters used in blockMeshDict File to mesh the geometry-(0.5 Page)

2.5 Boundary patches-This explains the different types of boundary patches that can be used in blockMeshDict File-(1-2 Pages)

2.6 Paraview-Visualization of the geometry-(1 Page)

3. Chapter 3: Creating curved geometry in OpenFOAM-(6-7 Pages)

3.1 Creating a curved geometry-This explains the steps for creating a curved geometry in OpenFOAM-(1 Page)

3.2 Flow over cylinder- Description of the geometry for 2-D flow over cylinder-(1 Page)

3.3 Breaking the geometry into blocks(0.5 Page)

3.4 BlockMeshDict file structure(0.5-1 Page)

3.5 Creating curved edges

3.5.1 Arcs for curved geometry-This explains how to calculation of co-ordinate points over curved edges-(0.5 Page)

3.5.2 How to calculate the intermediate point on the arc-(0.5 Page)

3.6 Paraview-Visualization of geometry-(0.5 Page)

3.7 Different views in Paraview-Description of different views in Paraview and their uses-(1-2 Pages)

4. Chapter 4: Simulating flow in a Lid Driven Cavity using OpenFOAM-(9-10 Pages)

4.1 Problem specification-Description of the flow and circulation formation at the corners of the geomtery-(1-2 Pages)

4.2 File structure of Lid Driven Cavity-Description of the set of case file required for solving this problem-(1 Page)

4.3 Boundary Conditions-Description of the boundary patches and the problem specifications at the boundary(1 Page)

4.4 Meshing the Geometry-Specification of the meshing parameters used in blockMeshDict File-(0.5 Page)

4.5 Visualization of the geometry in paraview-(0.5 Page)

4.6 Solver-

4.6.1 Definitions of the solver-(0.5 Page)

4.6.2 Description of the solver used to solve this problem-(1 Pages)

4.7 Simulating Flow-Working of the controlDict File based on the problem case-(0.5-1 Page)

4.8 Post-Processing:Paraview-Flow visualization in Streamline contour-(0.5 Page)

4.9 Plot U and V velocity in Paraview-plotting X and Y direction velocity over the flow domain-(1 Page)

4.10 Validate result with analytical data-(0.5 Page)

4.11 Exercise-(0.5 Page)

5. Chapter 5: Supersonic flow over a wedge using OpenFOAM-(12-14 Pages)

5.1 Supersonic Flow over a Wedge

5.1.1 Description of different types of flow based on their speed-(1 Page)

5.1.2 Oblique shock wave-(1-2 Pages)

5.1.3 Supersonic flow over a wedge-Illustration of the flow over wedge and the shock wave formation-(1-2 Pages)

5.2 Problem specification-elaboration of the problem statement considered for this simulation-(0.5 Page)

5.3 Case files-Description of the set of case files required for solving this problem-(1 Page)

5.4 Boundary Conditions-Description of the boundary patches and the problem specifications at the boundary(1-1.5 Page)

5.5 Meshing the Geometry-Specification of the meshing parameters used in blockMeshDict File-(0.5 Page)

5.6 Visualization of the geometry in paraview-(0.5 Page)

5.7 Compressible Flow Solver

5.7.1 Compressible flow-A brief description of compressible flow-(0.5 Page)

5.7.2 Description of the solver used to solve this problem-(1 Page)

5.8 Simulating Flow-Working of the controlDict File based on the problem case-(0.5 Page)

5.9 Post-Processing:Paraview-Flow visualization in Velocity contour-(0.5 Page)

5.10 Calculating Mach number

5.10.1 Mach Number-Description of the non-dimensional number and its function-(0.5 Page)

5.10.2 Calculating Mach Number in OpenFOAM-(0.5 Page)

5.11 Post-Processing:Paraview-Flow visualization in Mach contour-(0.5 Page)

5.12 Post-Processing:Paraview-Flow visualization in Velocity and Temperature contourand its explanation-(1 Page)

5.13 Validate result with analytical data-(0.5 Page)

5.14 Exercise-(0.5 Page)

6. Chapter 6: Two Dimensional Analysis of laminar flow through a channel-(14-15 Pages)

6.1 2-D channel Flow

6.1.1 Laminar Flow-Description of different types of flow based on Reynold Number-(1 Page)

6.1.2 Internal Flow-Description of flow through surfaces-(0.5 Page)

6.1.3 2-D Channel Flow-Description of 2 dimensional flow through channels-(1-2 Pages)

6.2 Problem specification-Elaboration of the problem statement considered for this simulation-(0.5 Page)

6.3 Case files-Description of the set of case files required for solving this problem-(1 Page)

6.4 Boundary Conditions-Description of the boundary patches and the problem specifications at the boundary(1-1.5 Page)

6.5 Meshing the Geometry-Specification of the meshing parameters used in blockMeshDict File-(0.5 Page)

6.6 Solver-

6.6.1 Definitions of the solver-(0.5 Page)

6.6.2 Description of the solver used to solve this problem-(1 Pages)

6.7 Simulating Flow-Working of the controlDict File based on the problem case-(0.5-1 Page)

6.8 Post-Processing:Paraview

6.8.1 Visualization of different contours in Paraview-(1-2 Pages)

6.8.1 Velocity Contour visualization and boundary layer formation-(1 Page)

6.8.3 Boundary layer visualization-Plotting of velocity along Y-axis at different locations in the channel-(1 Page)

6.9 Analytical Results- Calculation of the analytical results and description of the equations used -(1-2 Pages)

6.10 Validate Computational result with analytical results(1 Page)

6.11 Exercise-(0.5 Page)

7. Chapter 7: Turbulent flow in a lid driven cavity using OpenFOAM-(10-12 Pages)

7.1 Turbulent Flow-

7.1.1 Description of different types of flow based on Reynold Number-(1 Page)

7.1.2 Important characteristics of turbulent Flow-(1 Page)

7.2 Turbulent flow in a lid driven cavity- Description of the flow in lid driven cavity-(0.5 Page)

7.3 Problem specification-Elaboration of the problem statement considered for this simulation-(0.5 Page)

7.4 Case files-Description of the set of case files required for solving this problem-(1 Page)

7.5 Boundary Conditions-Description of the boundary patches and the problem specifications at the boundary(1 Page)

7.6 Meshing the Geometry-Specification of the meshing parameters used in blockMeshDict File-(0.5 Page)

7.7 Solver-

7.7.1 Definitions of the solver-(0.5 Page)

8.10 Post-Processing:Paraview

- 8.10.1 Visualization of different contours in Paraview-(1 Page)
- 8.10.2 Visualization of velocity contour and boundary layer formation on the flat plate surface -(1 Page)
- 8.10.3 Velocity plots to visualize the boundary layer velocity profile-(0.5 Page)
- 8.10.4 Velocity vector plot-Potting of velocity vector along the flow domain-(0.5 Page)

8.11 Exercise-Changing the grid spacing and comparing the results-(1 Page)