

Outline for CFD through Spoken Tutorials

Chapters in the book will be written based on this outline

9. Chapter 9: Simulating Hagen Poiseuille flow - (10 pages)

9.1. What is Hagen Poiseuille flow - 0.5 Page

9.1.1. Definition with diagram and expression for Pressure drop - 1.5 page

9.2. Dividing the pipe into blocks for blockMesh - 1 page

9.2.1. Creating o-grid for the pipe

9.3. Creating the blocks in blockMeshDict - this will explain the how to divide the pipe into blocks - 2 page

9.4. Selecting solver for the case - setting up 0, constant and system folder - 1 page

9.5. Paraview for visualization - Velocity and Pressure contours. Findind the maximum velocity at the center of the pipe - 3 page

9.6. Matching the velocity with analytical results - 0.5 page

9.7. Excercise - 0.5 page

10. Chapter 10: Downloading and Installing Salome - (5/6 Pages)

10.1. Introduction to Salome - 0.5 Page

10.2. Installation from Salome website

10.2.1. Steps to create a account on Salome website - 2 Page

10.2.2. Downloading Salome binaries - 1 Page

10.2.3. Creating a folder and installing Salome - 1 Page

10.3. How to start Salome - 0.5 Page

11. Chapter 11: Creating and Meshing a curved pipe geometry in Salome for OpenFOAM - (13 - 15 pages)

11.1. Problem definition for flow in a curved pipe - 1 Page

11.2. How to start salome for creating a geometry - Choosing the geometry module, dimensions (mm,cm,m) - 1 Page

11.3. Creating a curved pepe geometry in Salome

- 11.3.1. This will cover use of how to use 2D sketch feature for creating a circle - 1 Page
- 11.3.2. Creating face for the circle - 0.5 Page
- 11.3.3. Extruding the face - 0.5 Page
- 11.3.4. How to extrude the remaining faces to complete the curved pipe - 2 Pages
- 11.4. How to create groups to create sets for faces for boundary names - 1 Page
 - 11.4.1. Saving the geometry in *.hdf format - 0.5 Page
- 11.5. Using the Mesh module for Meshing the geometry - 1 Page
 - 11.5.1. Using 2D and 3D meshing algorithms - 0.5 Page
 - 11.5.2. Submesh utility to modify the mesh in the direction of the flow - submesh gives a better mesh control to capture the required flow physics - 1 Page
 - 11.5.3. Grouping the mesh - group mesh by color to identify the required boundary faces - 1 Page
- 11.6. Saving the Mesh files to be used in OpenFOAM - 0.5 Page
- 12. Chapter 12 : Exporting geometry from Salome to OpenFOAM - (12 Pages)**
 - 12.1. Using geometry created in the previous chapter - 1 Page
 - 12.2. Mesh module for Meshing the geometry and grouping the mesh - 2 Page
 - 12.3. Exporting the mesh file in *.unv format - 0.5 Page
 - 12.4. Setting up a case directory in OpenFOAM - 0.5 Page
 - 12.4.1. Choosing a solver in OpenFOAM - 0.5 Page
 - 12.4.2. Setting up case directory along with the mesh file - 1 Page
 - 12.4.3. Command to convert mesh file into OpenFOAM format - 1 Page
 - 12.5. Utility to scale down the geometry according to m, cm and mm. In case the user does not define the units we can scale it down using this utility in OpenFOAM - 1 Page
 - 12.6. Setting up pressure and velocity file names according to boundary names used in creating geometry - 1 Page
 - 12.7. Visualising the geomtry in Paraview - 2 Pages
- 13. Chapter 13 : Introduction to snappyHexMesh (8-9 Pages)**
 - 13.1. What is snappyHexmesh - utility for using CAD files (stl format) directly in OpenFOAM - 0.5 Page
 - 13.2. Basic steps required for using snappyHexMesh
 - 13.2.1. Creating a base mesh using blockMesh - 1 Page
 - 13.2.2. Refining the base mesh - 1 Page
 - 13.2.3. Removing the unused cells - 1 Page
 - 13.2.4. Snapping mesh to surface - 1 Page

- 13.2.5. Adding mesh layers to the surface - 1 Page
- 13.3. Using an example problem of Flange for snappyHexMesh - 0.5 Page
- 13.4. Explaining the parameters in snappyHexMeshDict file in systems folder - it is very important since this being an automated mesh generation we need to carefully define the various parameters for mesh generation - 3 Pages
- 14. Chapter 14 : Generating mesh using SnappyHexMesh - (15 Pages)**
 - 14.1. Using an example problem from the previous chapter for mesh generation , Flange - 1 Page
 - 14.2. Setting up a case directory - 0.5 Page
 - 14.3. How to use the stl file for flange available in the OpenFOAM tutorial directory - we need to set the path for the stl in triSurface directory of constant folder - 1 Page
 - 14.4. Setting up the blockMeshDict file for creating the base mesh - 1 Page
 - 14.5. Making changes in the snappyHexMeshDict file according to the geometry - castellated mesh, mesh layer addition, boundary layer, etc - 3/4 Pages
 - 14.6. Setting up the Pressure, Velocity and Temperature files in zero folder with addition of a patch for flange - 1 Page
 - 14.7. Steps for snappyHexmesh - 1 Page
 - 14.7.1. blockMesh - for base mesh creation - 0.5 Page
 - 14.7.2. surfaceFeatureExtract - extracting the surface features for the mesh - 1 page
 - 14.7.3. snappyHexMesh - for snapping the mesh according to the geometry. Also different flags used while using this command - 1 Page
 - 14.8. Using laplacianFoam for solving the case - what is laplacianFoam and governing equations - 1 Page
 - 14.9. Post-processing using Paraview showing mesh generated using snappyHexMesh, Temperature distribution in the Flange - 2 Page
 - 14.10. Example Problem - 0.5 Page
- 15. Chapter 15: Importing Mesh from Third Party Software in OpenFOAM - (12 Pages)**
 - 15.1. Why is the need to import mesh files in OpenFOAM - 0.5 Page
 - 15.2. Solving Flow over a square cylinder as an example problem
 - 15.2.1. Geometry for flow over square cylinder - 1 Page
 - 15.2.2. Mesh size used for the geometry - 1 Page
 - 15.2.3. Case directory creation - 1 Page
 - 15.3. Command for importing fluent mesh file in OpenFOAM - 0.5 Page
 - 15.4. How to change boundary names in constant folder - 1 Page

- 15.5. Using appropriate boundary conditions in 0 folder - 1 Page
- 15.6. Post-Processing in paraview - 3 Page
- 15.7. Commands to import mesh files from other third party softwares - 1 Page
- 15.8. Exercise Problem - 1 Page
- 16. Chapter 16 : Installing and Running Gmsh - (7 Pages)**
 - 16.1. Introduction to Gmsh - 0.5 Page
 - 16.2. Installation
 - 16.2.1. Download Gmsh from website - download a tar file according to 32/64 bit OS - 1 Page
 - 16.2.2. Install Gmsh from Synaptic Package Manager - 1 Page
 - 16.3. Creating a user directory for Gmsh - 0.5 Page
 - 16.4. Running Gmsh - double click on the Gmsh executable or type Gmsh in the terminal window - 1 Page
 - 16.5. Creating a basic geometry in Gmsh - Cube
 - 16.5.1. Points - 0.5 Page
 - 16.5.2. Lines - 0.5 Page
 - 16.5.3. Surfaces - 0.5 Page
 - 16.5.4. Volume - 0.5 Page
 - 16.6. Example Problem - 0.5 Page
- 17. Chapter 17 : Creating a Sphere in Gmsh (6 Pages)**
 - 17.1. Problem Definition - 1 Page
 - 17.2. Creating a Sphere in Gmsh - Define points for the sphere. Using Circular Arc feature in geometry module for creating a sphere - 1 Page
 - 17.3. Creating faces for the sphere - 1 Page
 - 17.4. Creating Volume for the Sphere - 1 Page
 - 17.5. Editing the geometry (geo) file in Gmsh to Control the mesh - 1 Page
 - 17.6. Exercise problem - 0.5 Page
- 18. Chapter 18 : Unstructured Mesh Generation using Gmsh**
 - 18.1. Using the Sphere generated in the previous chapter for creating mesh for flow over a sphere - 1 Page
 - 18.2. Problem Definition and boundary conditions - 1 Page
 - 18.3. Creating a rectangular domain for the sphere - 1 page
 - 18.3.1. Points - 0.5 Page
 - 18.3.2. lines - 0.5 Page
 - 18.3.3. Surface - 0.5 Page

- 18.3.4. Volume - 0.5 Page
- 18.4. Volume subtraction - subtract the volume of the sphere from the outer domain for meshing - 1 page
- 18.5. Adding Physical Surfaces and Volume for creating sets for faces - 2 Page
- 18.6. Adding boundary layer for sphere - 1 Page
- 18.7. Meshing the geometry using 1D, 2D and 3D mesh generation - 1 Page
- 18.8. Refining the mesh using Netgen optimization utility - 1 Page
- 18.9. Saving the mesh file for using it in OpenFOAM - 0.5 Page