Overview of OpenFOAM

Spoken Tutorial Project
https://spoken-tutorial.org
National Mission on Education through ICT
http://sakshat.ac.in/
Ashley Melvin & Swetha Sridhar
IIT Bombay

September 5, 2020

Learning Objectives

We will learn about:

Learning Objectives

We will learn about:

- `OpenFOAM` **and its capabilities**

Learning Objectives

We will learn about:

- `OpenFOAM` **and its capabilities**
- **Basic** `OpenFOAM` **utilities**

Learning Objectives

We will learn about:

- `OpenFOAM` **and its capabilities**
- **Basic** `OpenFOAM` **utilities**
- **The content available in various tutorials in this series**

Prerequisites

Prerequisites

- **You should have basic knowledge of** `computational fluid dynamics`

System Specifications

System Specifications

- **Ubuntu Linux OS version 18.04**

System Specifications

- **Ubuntu Linux OS version 18.04**
- **OpenFOAM version 7**

System Specifications

- Ubuntu Linux OS version 18.04
- OpenFOAM version 7
- ParaView version 5.6.0

System Specifications

- **Ubuntu Linux OS version 18.04**
- **OpenFOAM version 7**
- **ParaView version 5.6.0**
- **gedit Text Editor**

About OpenFOAM®

About OpenFOAM®

- Open source Field Operation And Manipulation

About OpenFOAM®

- Open source Field Operation And Manipulation
- **It is licensed under** GNU General Public Licence **by** OpenCFD Ltd.

About OpenFOAM®

Henceforth, in this series, whenever `OpenFOAM` **is mentioned it indicates** `OpenFOAMÂ®`

About OpenFOAM

- `Open source Computational Fluid Dynamics` **software**

About OpenFOAM

- `Open source Computational Fluid Dynamics` **software**
- **It is a** `CFD` **toolbox written in** `C++`

About OpenFOAM

- `Open source Computational Fluid Dynamics` **software**
- **It is a** `CFD` **toolbox written in** `C++`
- **It has an** `Object Oriented Programming` **interface**

About OpenFOAM

- `OpenFOAM` **is available for** `Linux`**,** `Mac` **and** `Windows` **operating systems**

OpenFOAM Capabilities

- `OpenFOAM` **is a** `Finite Volume` **based** `CFD` **software**

OpenFOAM Capabilities

- `OpenFOAM` **is a** `Finite Volume` **based** `CFD` **software**
- **It uses both**

OpenFOAM Capabilities

- OpenFOAM **is a** Finite Volume **based** CFD **software**
- **It uses both**
  - Structured

OpenFOAM Capabilities

- OpenFOAM **is a** Finite Volume **based** CFD **software**
- **It uses both**
  - Structured
  - Unstructured **grid**

Mesh Generation

- OpenFOAM **has an in-buit** mesh generation tool **called** blockMesh

Mesh Generation

- `OpenFOAM` **has an in-buit** `mesh generation tool` **called** `blockMesh`
- **It is used for** `structured meshing` **of simple geometries**

Mesh Generation

- `OpenFOAM` **has an in-buit** `mesh generation tool` **called** `blockMesh`
- **It is used for** `structured meshing` **of simple geometries**
- `OpenFOAM` **also has an advanced** `meshing tool` **called** `snappyHexMesh`

Mesh Conversion

Mesh Conversion

- ANSYS

Mesh Conversion

- ANSYS
- Fluent

Mesh Conversion

- ANSYS
- Fluent
- CFX

Mesh Conversion

- ANSYS
- Fluent
- CFX
- Gmsh

Mesh Conversion

- `ANSYS`
- `Fluent`
- `CFX`
- `Gmsh`
- `STAR-CD` **etc.**

Mesh Conversion

- **Imported** `mesh` **can be converted to the format** `OpenFOAM` **uses**

Mesh Conversion

- **Imported** `mesh` **can be converted to the format** `OpenFOAM` **uses**
- **For example, a** `fluent mesh` **is converted using the** `fluentMeshToFoam` **utility**

Solvers

Solvers

- `Incompressible flows`

Solvers

- `Incompressible flows`
- `Compressible flows`

Solvers

- Incompressible flows
- Compressible flows
- Multiphase flows

Solvers

- Incompressible flows
- Compressible flows
- Multiphase flows
- Heat transfer

Solvers

- Incompressible flows
- Compressible flows
- Multiphase flows
- Heat transfer
- Combustion systems

Solvers

- Incompressible flows
- Compressible flows
- Multiphase flows
- Heat transfer
- Combustion systems
- Molecular dynamics

Solvers

- Incompressible flows
- Compressible flows
- Multiphase flows
- Heat transfer
- Combustion systems
- Molecular dynamics
- Magnetohydrodynamic flows

ParaView

- **Visualizing the simulated results comes under the** `post-processing` **stage**

ParaView

- **Visualizing the simulated results comes under the** `post-processing` **stage**
- `ParaView` **is the most commonly used software for** `post-processing OpenFOAM` **results**

Summary

We have learnt about:

- `OpenFOAM` **and its capabilities**
- **Basic** `OpenFOAM` **utilities**
- **The content available in various tutorials in this series**

About the Spoken Tutorial Project

- Watch the video available at https://spoken-tutorial.org/What_is_a_Spoken_Tutorial
- It summarises the Spoken Tutorial project
- If you do not have good bandwidth, you can download and watch it

Spoken Tutorial Workshops

The Spoken Tutorial Project Team

- **Conducts workshops using spoken tutorials**
- **Gives certificates to those who pass an online test**
- **For more details, please write to contact@spoken-tutorial.org**

Spoken Tutorial Forum

- Questions in THIS Spoken Tutorial?
- Visit https://forums.spoken-tutorial.org/
- Choose the minute and second where you have the question
- Explain your question briefly
- The Spoken Tutorial project will ensure an answer

You will have to register to ask questions

FOSSEE Forum

- Questions not related to the Spoken Tutorial?
- Do you have general / technical questions on the Software?
- Please visit the FOSSEE Forum https://forums.fossee.in/
- Choose the Software and post your question

FOSSEE Case Study Project

- **The FOSSEE team coordinates solving feasible CFD problems of reasonable complexity using OpenFOAM**
- **We give honorarium and certificates to those who do this**
- **For more details, please visit:**
  https://cfd.fossee.in/
  https://fossee.in/

fossee better education

Acknowledgements

- **Spoken Tutorial Project is supported by the MHRD, Government of India**