Spoken Tutorial

Introduction to OpenFOAM

Talk to a Teacher

http://www.sakshat.ac.in

National Mission on Education through ICT

http://spoken-tutorial.org

Rahul Joshi and Saurabh Sawant

Date: August 13, 2019

Talk to a Teacher

Open Source Field Operation and Manipulation:

Talk to a Teacher

About OpenFOAM

Open Source Field Operation and Manipulation:

- is an Open Source Computational Fluid Dynamics Software

Talk to a Teacher

About OpenFOAM

Open Source Field Operation and Manipulation:

- **is an Open Source Computational Fluid Dynamics Software**

Talk to a Teacher

About OpenFOAM

Open Source Field Operation and Manipulation:

- **is an Open Source Computational Fluid Dynamics Software**

About OpenFOAM

CFD tool box:

Talk to a Teacher

About OpenFOAM

CFD tool box:

- written in C++ working on Linux operating systems

Talk to a Teacher

About OpenFOAM

CFD tool box:

- **written in C++ working on Linux operating systems**

Talk to a Teacher

About OpenFOAM

CFD tool box:

- **written in C++ working on Linux operating systems**

Talk to a Teacher

About OpenFOAM

OpenFOAM:

About OpenFOAM

OpenFOAM:

- **has an Object Oriented Programming Interface**

About OpenFOAM

OpenFOAM:

- **has an Object Oriented Programming Interface**

About OpenFOAM

OpenFOAM:

- has an Object Oriented Programming Interface
- It is licensed under GNU General Public Licence

Talk to a Teacher

About OpenFOAM

OpenFOAM:

- **has an Object Oriented Programming Interface**
- **It is licensed under GNU General Public Licence**

Talk to a Teacher

About OpenFOAM

OpenFOAM:

- **has an Object Oriented Programming Interface**
- **It is licensed under GNU General Public Licence**

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

OpenFOAM Capabilities

OpenFOAM:

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

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

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

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- **is a** *Finite Volume* **based CFD software**
- **using both structured and unstructured grid**

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- **is a** *Finite Volume* **based CFD software**
- **using both structured and unstructured grid**

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- **is a** *Finite Volume* **based CFD software**
- **using both structured and unstructured grid**

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- has a mesh generation tool called as blockMesh

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- **has a mesh generation tool called as blockMesh**

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- has a mesh generation tool called as blockMesh
  - It is useful for structured meshing ,easy and smaller grids

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- has a mesh generation tool called as blockMesh
  - It is useful for structured meshing ,easy and smaller grids

OpenFOAM Capabilities

OpenFOAM:

- **has a mesh generation tool called as blockMesh**
  - **It is useful for structured meshing ,easy and smaller grids**
  - **mesh input is in form of a script and not a GUI**

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- has a mesh generation tool called as blockMesh
  - It is useful for structured meshing ,easy and smaller grids
  - mesh input is in form of a script and not a GUI

Talk to a Teacher

OpenFOAM:

- has a mesh generation tool called as blockMesh
  - It is useful for structured meshing ,easy and smaller grids
  - mesh input is in form of a script and not a GUI
- It also has an Advanced Meshing tool called as : snappyHexMesh

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- has a mesh generation tool called as blockMesh
    - It is useful for structured meshing ,easy and smaller grids
    - mesh input is in form of a script and not a GUI
- It also has an Advanced Meshing tool called as : snappyHexMesh

Talk to a Teacher

OpenFOAM Capabilities

OpenFOAM:

- has a mesh generation tool called as blockMesh
  - It is useful for structured meshing ,easy and smaller grids
  - mesh input is in form of a script and not a GUI
- It also has an Advanced Meshing tool called as : snappyHexMesh

Talk to a Teacher

Importing Mesh Files

- **We can import mesh files into openFOAM using third party softwares like**

Importing Mesh Files

- **We can import mesh files into openFOAM using third party softwares like**

Talk to a Teacher

Importing Mesh Files

- **We can import mesh files into openFOAM using third party softwares like**
  - **fluentMeshToFoam**

Talk to a Teacher

Importing Mesh Files

- **We can import mesh files into openFOAM using third party softwares like**
  - **fluentMeshToFoam**

Importing Mesh Files

- We can import mesh files into openFOAM using third party softwares like
  - fluentMeshToFoam
  - cfxToFoam

Talk to a Teacher

Importing Mesh Files

- **We can import mesh files into openFOAM using third party softwares like**
  - **fluentMeshToFoam**
  - **cfxToFoam**

Importing Mesh Files

- We can import mesh files into openFOAM using third party softwares like
  - fluentMeshToFoam
  - cfxToFoam
- useful for large and complex geometries

Talk to a Teacher

Importing Mesh Files

- **We can import mesh files into openFOAM using third party softwares like**
  - **fluentMeshToFoam**
  - **cfxToFoam**
- **useful for large and complex geometries**

Talk to a Teacher

Importing Mesh Files

- **We can import mesh files into openFOAM using third party softwares like**
  - **fluentMeshToFoam**
  - **cfxToFoam**
- **useful for large and complex geometries**
- **arbitary polyhedral mesh**

Talk to a Teacher

Importing Mesh Files

- We can import mesh files into openFOAM using third party softwares like
  - fluentMeshToFoam
  - cfxToFoam
- useful for large and complex geometries
- arbitary polyhedral mesh

Talk to a Teacher

Importing Mesh Files

- We can import mesh files into openFOAM using third party softwares like
  - fluentMeshToFoam
  - cfxToFoam
- useful for large and complex geometries
- arbitary polyhedral mesh

Talk to a Teacher

Boundary Conditions

- **Various default boundary conditions which are available**

Talk to a Teacher

Boundary Conditions

- **Various default boundary conditions which are available**

Talk to a Teacher

Boundary Conditions

- **Various default boundary conditions which are available**
- **Users can also modify the boudnary conditions according to their case**

Boundary Conditions

- **Various default boundary conditions which are available**
- **Users can also modify the boudnary conditions according to their case**

Talk to a Teacher

Boundary Conditions

- **Various default boundary conditions which are available**
- **Users can also modify the boudnary conditions according to their case**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**
- **Multiphase flows (interFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**
- **Multiphase flows (interFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**
- **Multiphase flows (interFoam)**
- **Combustion (chemFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**
- **Multiphase flows (interFoam)**
- **Combustion (chemFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**
- **Multiphase flows (interFoam)**
- **Combustion (chemFoam)**
- **Particle-tracking flows (coalChemistryFoam)**

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**
- **Multiphase flows (interFoam)**
- **Combustion (chemFoam)**
- **Particle-tracking flows (coalChemistryFoam)**

Talk to a Teacher

Solvers

- Incompressible flows (icoFoam)
- Compressible flows (sonicFoam)
- Multiphase flows (interFoam)
- Combustion (chemFoam)
- Particle-tracking flows (coalChemistryFoam)
- Molecular Dynamics (mdFoam)

Talk to a Teacher

Solvers

- Incompressible flows (icoFoam)
- Compressible flows (sonicFoam)
- Multiphase flows (interFoam)
- Combustion (chemFoam)
- Particle-tracking flows (coalChemistryFoam)
- Molecular Dynamics (mdFoam)

Talk to a Teacher

Solvers

- Incompressible flows (icoFoam)
- Compressible flows (sonicFoam)
- Multiphase flows (interFoam)
- Combustion (chemFoam)
- Particle-tracking flows (coalChemistryFoam)
- Molecular Dynamics (mdFoam)
- MHD flows (mhdFoam), etc

Talk to a Teacher

Solvers

- **Incompressible flows (icoFoam)**
- **Compressible flows (sonicFoam)**
- **Multiphase flows (interFoam)**
- **Combustion (chemFoam)**
- **Particle-tracking flows (coalChemistryFoam)**
- **Molecular Dynamics (mdFoam)**
- **MHD flows (mhdFoam), etc**

Talk to a Teacher

Solvers

- Incompressible flows (icoFoam)
- Compressible flows (sonicFoam)
- Multiphase flows (interFoam)
- Combustion (chemFoam)
- Particle-tracking flows (coalChemistryFoam)
- Molecular Dynamics (mdFoam)
- MHD flows (mhdFoam), etc

Talk to a Teacher

Solvers

- **Users can create their own solvers**

Solvers

- **Users can create their own solvers**

Talk to a Teacher

Solvers

- **Users can create their own solvers**
- **Modify the existing solvers**

Talk to a Teacher

Solvers

- **Users can create their own solvers**
- **Modify the existing solvers**

Talk to a Teacher

Solvers

- **Users can create their own solvers**
- **Modify the existing solvers**

Talk to a Teacher

- **Parallel processing is easy in OpenFOAM and it supports Open MPI**

Talk to a Teacher

Parallel Processing

- **Parallel processing is easy in OpenFOAM and it supports Open MPI**

Talk to a Teacher

Parallel Processing

- **Parallel processing is easy in OpenFOAM and it supports Open MPI**
- **We can use 'n' number of processors**

Talk to a Teacher

Parallel Processing

- **Parallel processing is easy in OpenFOAM and it supports Open MPI**
- **We can use 'n' number of processors**

Talk to a Teacher

Parallel Processing

- **Parallel processing is easy in OpenFOAM and it supports Open MPI**
- **We can use 'n' number of processors**

Talk to a Teacher

Post Processing

- **OpenFOAM results can be visualized using Paraview which is an open source software**

Talk to a Teacher

Post Processing

- **OpenFOAM results can be visualized using Paraview which is an open source software**

Talk to a Teacher

Post Processing

- **OpenFOAM results can be visualized using Paraview which is an open source software**
- **OpenFOAM results can also be visulalised in other softwares like**

Talk to a Teacher

Post Processing

- **OpenFOAM results can be visualized using Paraview which is an open source software**
- **OpenFOAM results can also be visulalised in other softwares like**

Talk to a Teacher

Post Processing

- **OpenFOAM results can be visualized using Paraview which is an open source software**
- **OpenFOAM results can also be visulalised in other softwares like**
  - **Tecplot**

Talk to a Teacher

Post Processing

- OpenFOAM results can be visualized using Paraview which is an open source software
- OpenFOAM results can also be visualised in other softwares like
    - Tecplot

Talk to a Teacher

Post Processing

- **OpenFOAM results can be visualized using Paraview which is an open source software**
- **OpenFOAM results can also be visulalised in other softwares like**
  - **Tecplot**
  - **Ensight**

Talk to a Teacher

Post Processing

- OpenFOAM results can be visualized using Paraview which is an open source software
- OpenFOAM results can also be visualised in other softwares like
  - Tecplot
  - Ensight

Talk to a Teacher

Post Processing

- OpenFOAM results can be visualized using Paraview which is an open source software
- OpenFOAM results can also be visualised in other softwares like
  - Tecplot
  - Ensight

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**
- **Solver capabilities of OpenFOAM are similar to that of**

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**
- **Solver capabilities of OpenFOAM are similar to that of**

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**
- **Solver capabilities of OpenFOAM are similar to that of**
    - **Fluent**

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**
- **Solver capabilities of OpenFOAM are similar to that of**
  - **Fluent**

Talk to a Teacher

Equivalent to commercial software

- Commercial softwares are costly and source code is not available
- Solver capabilities of OpenFOAM are similar to that of
  - Fluent
  - CFX

Talk to a Teacher

Equivalent to commercial software

- Commercial softwares are costly and source code is not available
- Solver capabilities of OpenFOAM are similar to that of
  - Fluent
  - CFX

Talk to a Teacher

Equivalent to commercial software

- Commercial softwares are costly and source code is not available
- Solver capabilities of OpenFOAM are similar to that of
  - Fluent
  - CFX
  - StarCCm+

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**
- **Solver capabilities of OpenFOAM are similar to that of**
  - **Fluent**
  - **CFX**
  - **StarCCm+**

Talk to a Teacher

Equivalent to commercial software

- **Commercial softwares are costly and source code is not available**
- **Solver capabilities of OpenFOAM are similar to that of**
  - **Fluent**
  - **CFX**
  - **StarCCm+**

Talk to a Teacher

Modelling in OpenFOAM

Solver syntax in OpenFOAM:

Talk to a Teacher

Modelling in OpenFOAM

Solver syntax in OpenFOAM:

- is similar to that used in writing a Partial Differential Equation

Talk to a Teacher

Modelling in OpenFOAM

Solver syntax in OpenFOAM:

- **is similar to that used in writing a Partial Differential Equation**

Talk to a Teacher

Solver syntax in OpenFOAM:

- **is similar to that used in writing a Partial Differential Equation**

Talk to a Teacher

Forum to answer questions

- **Do you have questions on THIS Spoken Tutorial?**

Talk to a Teacher

Forum to answer questions

- **Do you have questions on THIS Spoken Tutorial?**
- **Choose the minute and second where you have the question.**

Talk to a Teacher

Forum to answer questions

- Do you have questions on THIS Spoken Tutorial?
- Choose the minute and second where you have the question.
- Explain your question briefly.

Talk to a Teacher

Forum to answer questions

- **Do you have questions on THIS Spoken Tutorial?**
- **Choose the minute and second where you have the question.**
- **Explain your question briefly.**
- **Someone from the FOSSEE team will answer them. Please visit**

Talk to a Teacher

Forum to answer questions

- **Questions not related to the Spoken Tutorial?**
- **Do you have general / technical questions on the Software?**
- **Please visit the FOSSEE Forum**

  http://forums.fossee.in/
- **Choose the Software and post your question.**

Talk to a Teacher

Case Study Project

- The FOSSEE team coordinates solving past, current or new CFD projects using OpenFOAM
- We give honorarium and certificate to those who do this

For more details, please visit this site:

http://cfd.fossee.in/

Talk to a Teacher

Acknowledgements

- Spoken Tutorial Project is a part of the Talk to a Teacher project
- It is supported by the National Mission on Education through ICT, MHRD, Government of India
- More information on this Mission is available at:

http://spoken-tutorial.org/NMEICT-Intro

Talk to a Teacher

Acknowledgements

- Spoken Tutorial Project is a part of the Talk to a Teacher project
- It is supported by the National Mission on Education through ICT, MHRD, Government of India
- More information on this Mission is available at http://spoken-tutorial.org/NMEICT-Intro

Talk to a Teacher

About the Contributor

- **Rahul Joshi - IIT BOMBAY signing off.**
- **Thanks for joining.**

Talk to a Teacher