

Spoken Tutorial

Installing and Running OpenFOAM and ParaView

Talk to a Teacher

<http://www.sakshat.ac.in>

National Mission on Education through ICT

<http://spoken-tutorial.org>

Rahul Joshi

Date: January 25, 2012



Learning Objectives

How to Install and Run:



Talk to a Teacher

Learning Objectives

How to Install and Run:

- OpenFOAM



Talk to a Teacher

Learning Objectives

How to Install and Run:

- OpenFOAM
- Paraview



Talk to a Teacher

Learning Objectives

How to Install and Run:

- **OpenFOAM**
- **Paraview**
- **Solve Lid Driven Cavity case**



Talk to a Teacher

System Requirement

- **Linux Operating System Ubuntu version 10.04**



Talk to a Teacher

System Requirement

- **Linux Operating System Ubuntu version 10.04**
- **OpenFOAM version 2.1.0**



Talk to a Teacher

System Requirement

- **Linux Operating System Ubuntu version 10.04**
- **OpenFOAM version 2.1.0**
- **ParaView version 3.12.0**



Talk to a Teacher

System Requirement

- Linux Operating System Ubuntu version 10.04
- OpenFOAM version 2.1.0
- ParaView version 3.12.0
- OpenFOAM does not run on Windows platform



Talk to a Teacher

System Requirement

- Linux Operating System Ubuntu version 10.04
- OpenFOAM version 2.1.0
- ParaView version 3.12.0
- OpenFOAM does not run on Windows platform
- OpenFOAM is compatible with Linux 10.04 and above



Talk to a Teacher

Prerequisites

Basic Knowledge of



Talk to a Teacher

Prerequisites

Basic Knowledge of

- Computational Fluid Dynamics



Talk to a Teacher

Prerequisites

Basic Knowledge of

- Computational Fluid Dynamics
- Linux Commands



Talk to a Teacher

Introduction to OpenFOAM

- It is an Open Source Computational Fluid Dynamics (CFD) Software



Talk to a Teacher

Introduction to OpenFOAM

- It is an Open Source Computational Fluid Dynamics (CFD) Software
- It contains a wide range of solvers for solving CFD problems of



Talk to a Teacher

Introduction to OpenFOAM

- It is an Open Source Computational Fluid Dynamics (CFD) Software
- It contains a wide range of solvers for solving CFD problems of
 - 2 Dimensions



Talk to a Teacher

Introduction to OpenFOAM

- It is an Open Source Computational Fluid Dynamics (CFD) Software
- It contains a wide range of solvers for solving CFD problems of
 - 2 Dimensions
 - 3 Dimensions



Talk to a Teacher

Introduction to ParaView

- It is used for visualizing results obtained in OpenFOAM



Talk to a Teacher

Installing OpenFOAM and Paraview

- **OpenFOAM and Paraview - Synaptic Package Manager**



Talk to a Teacher

Installing OpenFOAM and Paraview

- OpenFOAM and Paraview - Synaptic Package Manager
- Spoken tutorial on **Synaptic Package Manager**



Talk to a Teacher

Installing OpenFOAM and Paraview

- **OpenFOAM and Paraview - Synaptic Package Manager**
- **Spoken tutorial on Synaptic Package Manager**
- <http://spoken-tutorial.org/>



Talk to a Teacher

Downloading OpenFOAM and Paraview

- **Download and install OpenFOAM and ParaView from:**



Talk to a Teacher

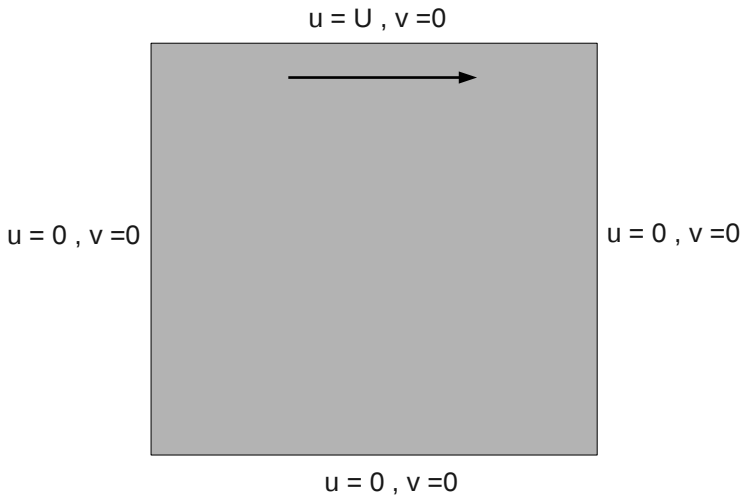
Downloading OpenFOAM and Paraview

- Download and install OpenFOAM and ParaView from:
- www.openfoam.com/download



Talk to a Teacher

Running OpenFOAM



(All velocities are in m/s)



Talk to a Teacher

Running OpenFOAM

- **Lid Driven Cavity**



Talk to a Teacher

Running OpenFOAM

- **Lid Driven Cavity**
- **Incompressible Solver : icoFoam**



Talk to a Teacher

Summary

- In this tutorial we learnt :



Talk to a Teacher

Summary

- In this tutorial we learnt :
 - How to install OpenFOAM and Paraview : Synaptic Package Manager and Website



Talk to a Teacher

Summary

- In this tutorial we learnt :
 - How to install OpenFOAM and Paraview : Synaptic Package Manager and Website
 - Solved a Lid Driven Cavity



Talk to a Teacher

Assignment

- **Install OpenFOAM and Paraview**



Talk to a Teacher

About the Spoken Tutorial Project

- Watch the video available at



Talk to a Teacher

About the Spoken Tutorial Project

- Watch the video available at http://spoken-tutorial.org/What_is_a_Spoken_Tutorial



Talk to a Teacher

About the Spoken Tutorial Project

- Watch the video available at http://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



Talk to a Teacher

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, contact sptutemail@gmail.com



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