

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



Talk to a Teacher

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



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

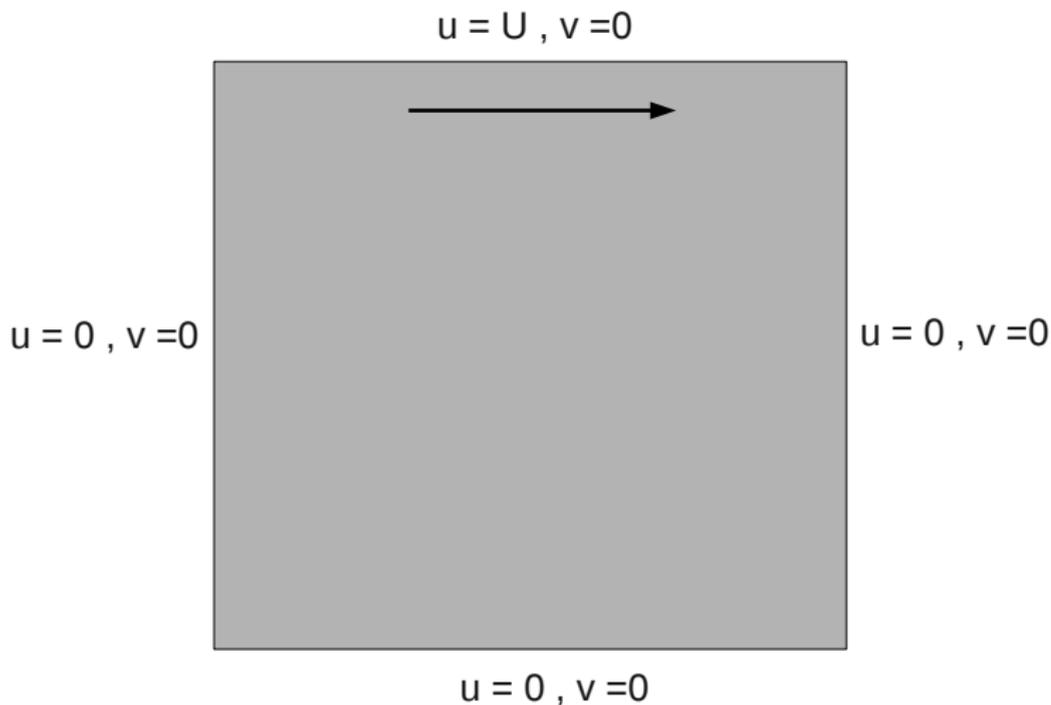
Downloading OpenFOAM and Paraview

- **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

- **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



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



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