# Simulating 2D laminar flow in a channel using OpenFOAM

Talk to a Teacher

http://spoken-tutorial.org

National Mission on Education through ICT

http://sakshat.ac.in

Rahul Joshi

Date: November 22, 2012

# Learning Objectives

- **2D Geometry of Channel**

# Learning Objectives

- **2D Geometry of Channel**

# Learning Objectives

- **2D Geometry of Channel**
- **Meshing the geometry**

# Learning Objectives

- **2D Geometry of Channel**
- **Meshing the geometry**

# Learning Objectives

- **2D Geometry of Channel**
- **Meshing the geometry**
- **Solving and post-processing results in Paraview**

# Learning Objectives

- **2D Geometry of Channel**
- **Meshing the geometry**
- **Solving and post-processing results in Paraview**

# Learning Objectives

- 2D Geometry of Channel
- Meshing the geometry
- Solving and post-processing results in Paraview
- Validation using analytic result

# System Requirement

- **Linux Operating System Ubuntu version 12.04**

# System Requirement

- **Linux Operating System Ubuntu version 12.04**

# System Requirement

- **Linux Operating System Ubuntu version 12.04**
- **OpenFOAM version 2.1.1**

# System Requirement

- **Linux Operating System Ubuntu version 12.04**
- **OpenFOAM version 2.1.1**

# System Requirement

- **Linux Operating System Ubuntu version 12.04**
- **OpenFOAM version 2.1.1**
- **ParaView version 3.12.0**

# System Requirement

- **Linux Operating System Ubuntu version 12.04**
- **OpenFOAM version 2.1.1**
- **ParaView version 3.12.0**
- **OpenFOAM 2.1.1 is supported on Ubuntu 12.04**

# System Requirement

- **The tutorials were recorded using the versions specified in previous slide.**
- **Subsequently the tutorials were edited to latest versions.**
- **To install latest system requirements go to Installation Sheet.**

# Prerequisite

- **Knowledge of how to create geometry using OpenFOAM**

# Prerequisite

- **Knowledge of how to create geometry using OpenFOAM**
- **If not, please refer to the relevant tutorials on http://spoken-tutorial.org**

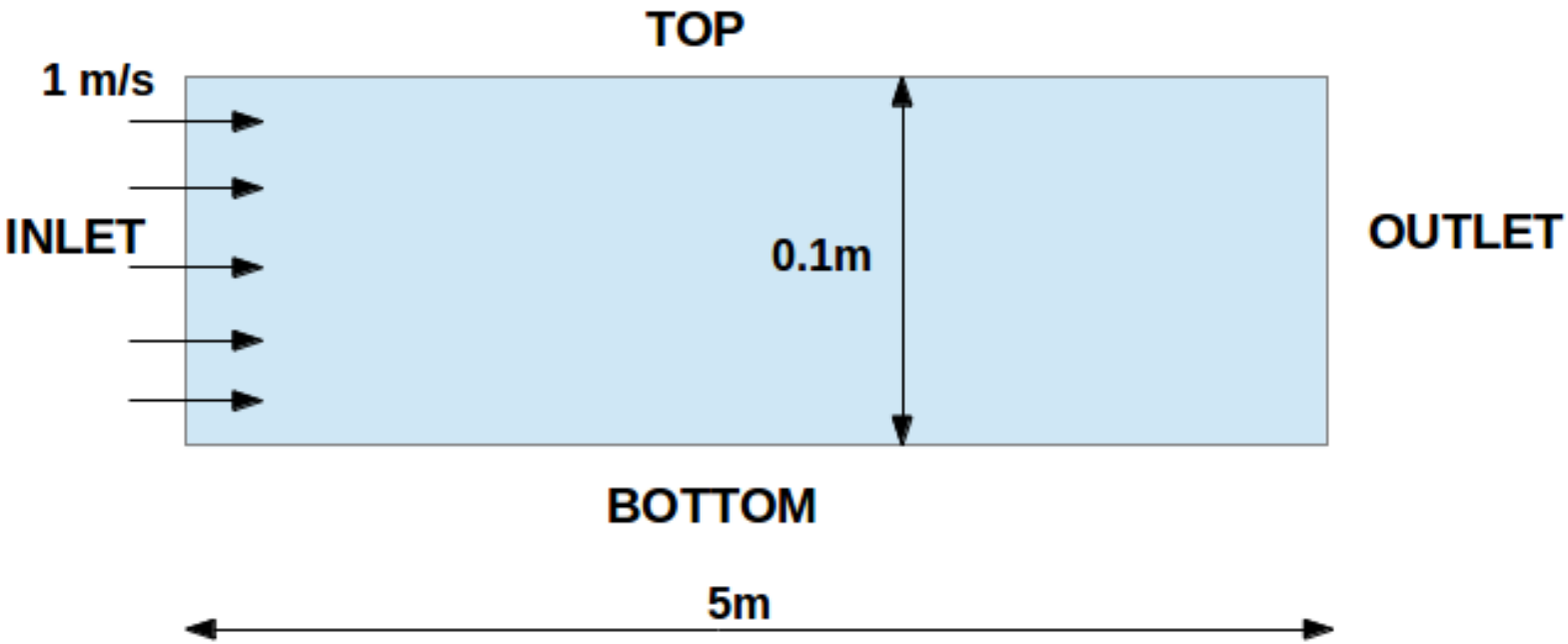
# About flow in channel

- **We simulate flow in a Channel to determine**

# About flow in channel

- **We simulate flow in a Channel to determine**
- **Flow development length along the downstream**

# Channel flow



# Boundary Conditions

- **Flow development length is given by the formula L=0.05*Re*D**

# Boundary Conditions

- **Flow development length is given by the formula L=0.05*Re*D**

# Boundary Conditions

- **Flow development length is given by the formula L=0.05*Re*D**
- **Channel is of length 5m and height 1m**

# Boundary Conditions

- **Flow development length is given by the formula L=0.05*Re*D**
- **Channel is of length 5m and height 1m**

# Boundary Conditions

- **Flow development length is given by the formula L=0.05*Re*D**
- **Channel is of length 5m and height 1m**
- **Inlet velocity is 1m/s**

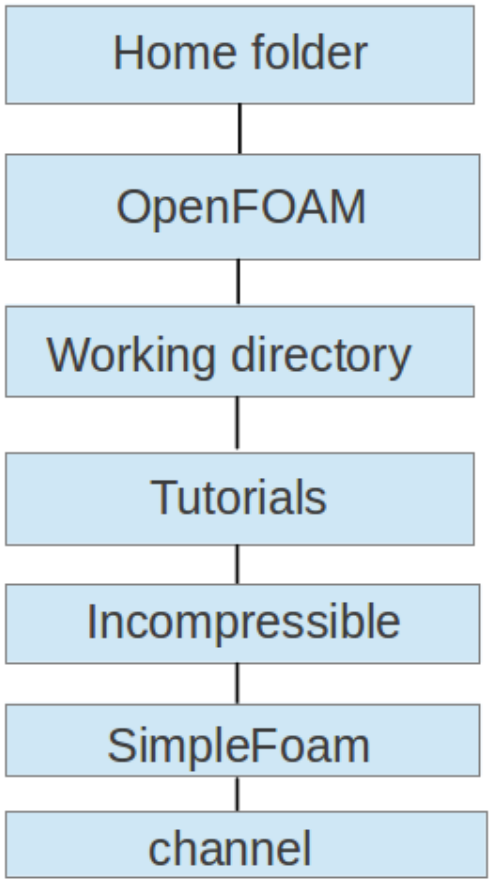
# Boundary Conditions

- **Flow development length is given by the formula L=0.05*Re*D**
- **Channel is of length 5m and height 1m**
- **Inlet velocity is 1m/s**

# Boundary Conditions

- **Flow development length is given by the formula L=0.05*Re*D**
- **Channel is of length 5m and height 1m**
- **Inlet velocity is 1m/s**
- **Reynolds number(Re)=100**

# File structure




Talk to a Teacher

# Calculate k

$$k = \frac{(U'^2x + U'^2y + U'^2z)}{2}$$

- 'k' is turbulent kinetic energy

# Calculate k

$$k = \frac{(U'^2x + U'^2y + U'^2z)}{2}$$

- **'k' is turbulent kinetic energy**

# Calculate k

$$k = \frac{(U'^2x + U'^2y + U'^2z)}{2}$$

- 'k' is turbulent kinetic energy
- $U'_x, U'_y, U'_z$ are velocity components

# Calculate k

$$k = \frac{(U'^2x + U'^2y + U'^2z)}{2}$$

- 'k' is turbulent kinetic energy
- $U'_x, U'_y, U'_z$ are velocity components

# Calculate k

$$k = \frac{(U'^2x+U'^2y+U'^2z)}{2}$$

- 'k' is turbulent kinetic energy
- $U'_x$,$U'_y$,$U'_z$ are velocity components
- $U'=0.05*u_{actual}$

# Calculate epsilon

$$\varepsilon = \frac{C^{0.75}\mu * k^{1.5}}{l}$$

- $\epsilon$ is rate of dissipation of turbulent energy

# Calculate epsilon

$$\varepsilon = \frac{C^{0.75}\mu * k^{1.5}}{l}$$

- $\epsilon$ **is rate of dissipation of turbulent energy**

# Calculate epsilon

$$\varepsilon = \frac{C^{0.75}{}_{\mu} * k^{1.5}}{l}$$

- $\epsilon$ **is rate of dissipation of turbulent energy**
- $C_{\mu}$ **is a constant and its value is 0.09**

# Calculate epsilon

$$\varepsilon = \frac{C^{0.75}{}_{\mu} * k^{1.5}}{l}$$

- $\epsilon$ **is rate of dissipation of turbulent energy**
- $\mathbf{C}_{\mu}$ **is a constant and its value is 0.09**

# Calculate epsilon

$$\varepsilon = \frac{C^{0.75}{}_{\mu} * k^{1.5}}{l}$$

- $\epsilon$ **is rate of dissipation of turbulent energy**
- $\mathbf{C}_{\mu}$ **is a constant and its value is 0.09**
- **'l' is the length of the pipe**

# Solver

- **SimpleFoam**

# Solver

- **SimpleFoam**
  - **Steady-state solver for incompressible and turbulent flows**

# Analytical result

- **Analytical result $U_{max}$ =1.5*$U_{avg}$**

# Analytical result

- **Analytical result $U_{max}$ =1.5*$U_{avg}$**

# Analytical result

- **Analytical result $U_{max}$ =1.5*$U_{avg}$**
- **OpenFOAM result $U_{max}$ = 1.48 m/s**

# Summary

- **File structure of channel**

# Summary

- **File structure of channel**

# Summary

- **File structure of channel**
- **Obtained solution using steady state solver**

# Summary

- **File structure of channel**
- **Obtained solution using steady state solver**

# Summary

- File structure of channel
- Obtained solution using steady state solver
- Viewed geometry in paraview

# Summary

- File structure of channel
- Obtained solution using steady state solver
- Viewed geometry in paraview

# Summary

- File structure of channel
- Obtained solution using steady state solver
- Viewed geometry in paraview
- Validation with analytical result

# Assignment

- **Solve the problem for Reynolds Number (Re) = 1500**

# Assignment

- **Solve the problem for Reynolds Number (Re) = 1500**

# Assignment

- **Solve the problem for Reynolds Number (Re) = 1500**
- **Validate with anlytical results**

# 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,please write to contact@spoken-tutorial.org**

# 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**

**http://forums.spoken-tutorial.org/**

# 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.**

# Lab Migration Project

- **We coordinate migration from commercial CFD software like ANSYS to OpenFOAM**
- **We conduct free Workshops and provide solutions to CFD Problem Statements in OpenFOAM**

**For more details, please visit this site:**

**http://cfd.fossee.in/**

# 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/**

# 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