Flow over a Flat plate using OpenFOAM

Talk to a Teacher

http://spoken-tutorial.org

National Mission on Education through ICT

http://sakshat.ac.in

Rahul Joshi

Date: November 19, 2012

Learning Objectives

- **Geometry of flat plate**

Learning Objectives

- **Geometry of flat plate**

Learning Objectives

- **Geometry of flat plate**
- **Changing the grid spacing in meshing**

Learning Objectives

- **Geometry of flat plate**
- **Changing the grid spacing in meshing**

Learning Objectives

- **Geometry of flat plate**
- **Changing the grid spacing in meshing**
- **Post processing results in ParaView**

Learning Objectives

- **Geometry of flat plate**
- **Changing the grid spacing in meshing**
- **Post processing results in ParaView**

Learning Objectives

- **Geometry of flat plate**
- **Changing the grid spacing in meshing**
- **Post processing results in ParaView**
- **Visualizing using Vector plot**

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**

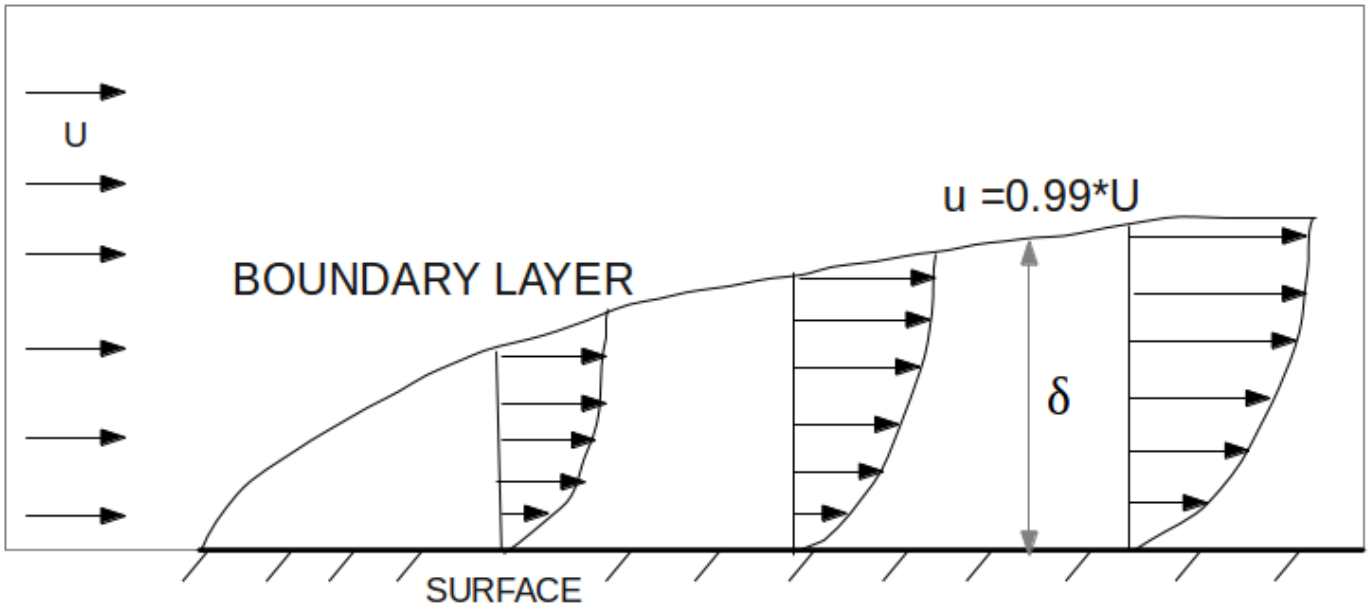
System Requirement

- **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.**

About Flow over flat plate

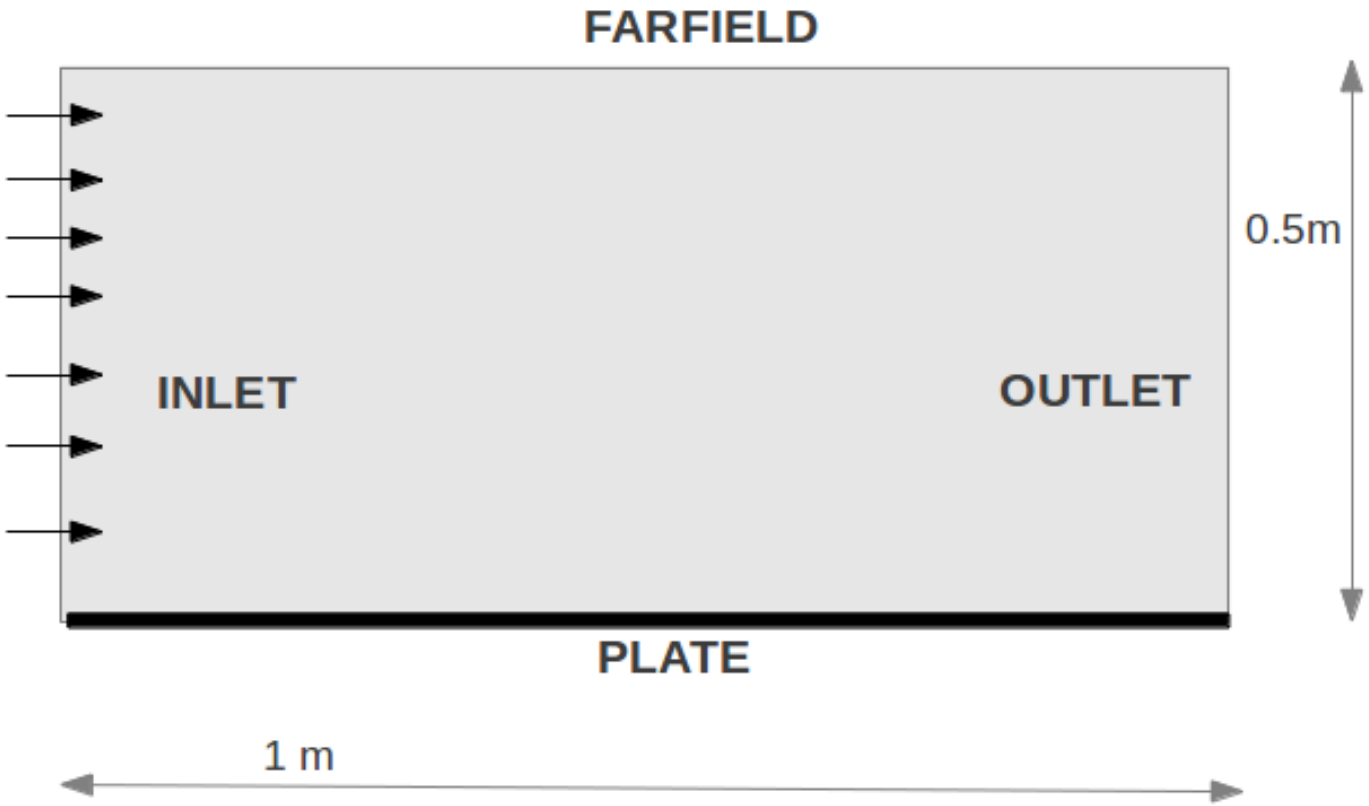
- **Fundamental problem in fluid mechanics**

Flow over Flat Plate



δ - Boundary layer height

Boundary conditions



Inlet parameters

- **Free stream velocity, U= 1m/s**

Inlet parameters

- **Free stream velocity, U= 1m/s**

Inlet parameters

- **Free stream velocity, U= 1m/s**
- **We are solving this for a Reynolds no, Re = 100**

Solver

- **simpleFoam**

Solver

- **simpleFoam**
  - **Steady state Solver for Incompressible**

Solver

- **simpleFoam**
  - **Steady state Solver for Incompressible**
  - **and turbulent flows**

Summary

- **In this tutorial we learnt**

Summary

- **In this tutorial we learnt**

Summary

- **In this tutorial we learnt**
  - **Gemetry and meshing of flat plate geometry**

Summary

- **In this tutorial we learnt**
  - **Gemetry and meshing of flat plate geometry**

Summary

- **In this tutorial we learnt**
  - **Gemetry and meshing of flat plate geometry**
  - **Vector plotting in paraview**

Assignment

- **Change the grid size as well as grid spacing**

Assignment

- **Change the grid size as well as grid spacing**

Assignment

- **Change the grid size as well as grid spacing**
- **Visualise using vector plots**

About the Spoken Tutorial Project

- **Watch the video available at**

About the Spoken Tutorial Project

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

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**

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