

# Simulating Hagen Poiseuille Flow through a Pipe

**Spoken Tutorial Project**

**<https://spoken-tutorial.org>**

**National Mission on Education through ICT**

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

**Divyesh Variya & Swetha Sridhar**

**IIT Bombay**

**June 30, 2020**



# Learning Objectives

**We will learn to:**



# Learning Objectives

**We will learn to:**

- ▶ **Set up the boundary and initial conditions**



# Learning Objectives

**We will learn to:**

- ▶ **Set up the** boundary and initial conditions
- ▶ **Set up the** physical Properties





# Learning Objectives

- ▶ **Set up the solve & write control** parameter
- ▶ **Run the** simulation



# System Specifications



# System Specifications

## ► Ubuntu Linux OS version 18.04



# System Specifications

- ▶ **Ubuntu Linux OS version 18.04**
- ▶ **OpenFOAM version 7**



# System Specifications

- ▶ **Ubuntu Linux OS version 18.04**
- ▶ **OpenFOAM version 7**
- ▶ **ParaView version 5.6.0**



# System Specifications

- ▶ **Ubuntu Linux OS version 18.04**
- ▶ **OpenFOAM version 7**
- ▶ **ParaView version 5.6.0**
- ▶ **gedit Text editor**



# Prerequisites



# Prerequisites

- ▶ **You should be familiar with** Hagen Poiseuille flow **& basic** Fluid Dynamics **concepts**



# Prerequisites

- ▶ **You should be familiar with** Hagen Poiseuille flow **& basic** Fluid Dynamics **concepts**
- ▶ **You should also be familiar with** **creation of a** 3D pipe Geometry **in** OpenFOAM



# Prerequisites

- ▶ If not, please go through the prerequisite `OpenFOAM` tutorial on <https://spoken-tutorial.org>



# Code Files

- ▶ **The files used in this tutorial are available in the Code Files link on this tutorial page**



# Code Files

- ▶ **The files used in this tutorial are available in the Code Files link on this tutorial page**
- ▶ **Please download and extract them**

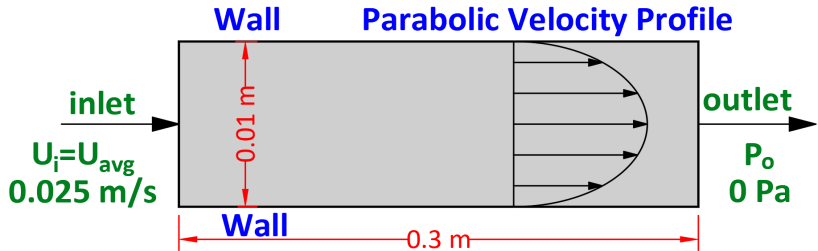


# Code Files

- ▶ **The files used in this tutorial are available in the `Code Files` link on this tutorial page**
- ▶ **Please download and extract them**
- ▶ **Make a copy and then use them while practising**



# Problem Setup



For water,

$\mu$  = Dynamic Viscosity =  $1\text{e-}03$

$\eta$  = Kinematic Viscosity =  $1\text{e-}06$

# Geometry Check



# Geometry Check

► Reynolds Number:

$$\text{► } Re = \frac{U_{avg} D}{\nu} = 250$$



# Geometry Check

► Reynolds Number:

►  $Re = \frac{U_{avg} D}{\nu} = 250$

► **Hence, the flow is** laminar



# Geometry Check

► Reynolds Number:

►  $Re = \frac{U_{avg} D}{\nu} = 250$

► Hence, the flow is laminar

► **Calculate** Entrance Length:



# Geometry Check

► Reynolds Number:

$$\text{► } Re = \frac{U_{avg} D}{\nu} = 250$$

► Hence, the flow is laminar

► Calculate Entrance Length:

$$\text{► } L_e = 0.06 * Re * D$$



# Geometry Check

- ▶ Reynolds Number:

- ▶  $Re = \frac{U_{avg} D}{\nu} = 250$

- ▶ **Hence, the flow is** laminar

- ▶ **Calculate** Entrance Length:

- ▶  $L_e = 0.06 * Re * D$

- ▶  $L_e = 0.15m$



# Solver detail

- ▶ `icoFoam` **is a** transient solver **for** incompressible, laminar flow **of** Newtonian fluids

**This is the most** suitable solver **for** Hagen Poiseuille flow



# Pressure Boundary Condition



# Pressure Boundary Condition

► **inlet: Zero Gradient**



# Pressure Boundary Condition

- ▶ **inlet: Zero Gradient**
- ▶ **outlet: Fixed Pressure (0 Pa)**



# Pressure Boundary Condition

- ▶ **inlet: Zero Gradient**
- ▶ **outlet: Fixed Pressure (0 Pa)**
- ▶ **wall: Zero Gradient**



# Velocity Boundary Condition



# Velocity Boundary Condition

- ▶ **inlet: Fixed Velocity ( $0.025 \text{ m/s}$ )**



# Velocity Boundary Condition

- ▶ **inlet: Fixed Velocity ( $0.025 \text{ m/s}$ )**
- ▶ **outlet: Zero Gradient**



# Velocity Boundary Condition

- ▶ **inlet: Fixed Velocity ( $0.025 \text{ m/s}$ )**
- ▶ **outlet: Zero Gradient**
- ▶ **wall: No Slip ( $0 \text{ m/s}$  velocity)**



# Formula and Analytical Solution



# Formula and Analytical Solution

## ► Pressure Drop along the pipe:

$$► P_i - P_o = \frac{32\mu U_{avg} L}{D^2}$$



# Formula and Analytical Solution

## ► Pressure Drop along the pipe:

$$► P_i - P_o = \frac{32\mu U_{avg} L}{D^2}$$

$$► P_i = 2.4 Pa$$



# Formula and Analytical Solution

## ▶ Pressure Drop along the pipe:

$$\text{▶ } P_i - P_o = \frac{32\mu U_{avg} L}{D^2}$$

$$\text{▶ } P_i = 2.4 Pa$$

## ▶ Maximum Velocity:

$$\text{▶ } U_{max} = 2 * U_{avg} = 0.05 m/s$$



# Result Comparison



# Result Comparison

	Analytical	OpenFOAM
<b>Velocity</b>	<b>0.05</b>	<b>0.05</b>
<b>Pressure</b>	<b>2.4 Pa</b>	<b>2.8 Pa</b>



# Result Comparison

	Analytical	OpenFOAM
Velocity	0.05	0.05
Pressure	2.4 Pa	2.8 Pa

- ▶ In CFD, there are always some deviations in results of analytical **and** numerical solutions



# Summary

**We have learnt to:**

- ▶ **Set up the boundary and initial conditions**
- ▶ **Set up the physical properties**



# Summary

- ▶ **Set up the solve & write control** parameter
- ▶ **Run the** simulation



# About the Spoken Tutorial Project

- ▶ Watch the video available at [https://spoken-tutorial.org/What\\_is\\_a\\_Spoken\\_Tutorial](https://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](mailto:contact@spoken-tutorial.org)



# Spoken Tutorial Forum

- ▶ **Questions in THIS Spoken Tutorial?**
- ▶ **Visit** <https://forums.spoken-tutorial.org/>
- ▶ **Choose the minute and second where you have the question**
- ▶ **Explain your question briefly**
- ▶ **The Spoken Tutorial project will ensure an answer**

**You will have to register to ask questions**



# FOSSEE Forum

- ▶ Questions not related to the Spoken Tutorial?
- ▶ Do you have general / technical questions on the Software?
- ▶ Please visit the FOSSEE Forum <https://forums.fossee.in/>
- ▶ Choose the Software and post your question



# FOSSEE Case Study Project

- ▶ **The FOSSEE team coordinates solving feasible CFD problems of reasonable complexity using OpenFOAM**
- ▶ **We give honorarium and certificates to those who do this**
- ▶ **For more details, please visit:**  
<https://cfd.fossee.in/>  
<https://fossee.in/>



# Acknowledgements

- ▶ **Spoken Tutorial Project is supported by the MHRD, Government of India**

