Simulation of Turbulent Flow in a 2D Channel

Spoken Tutorial Project
https://spoken-tutorial.org
National Mission on Education through ICT
http://sakshat.ac.in/

Padmini Priyadarshini & Swetha Sridhar
IIT Bombay

20 March 2020

Learning Objectives

We will learn how to:

Learning Objectives

We will learn how to:

- **Implement** `k-epsilon model`

Learning Objectives

We will learn how to:

- **Implement** `k-epsilon model`
- **Set up** `turbulence parameters`

Learning Objectives

We will learn how to:

- **Implement** `k-epsilon model`
- **Set up** `turbulence parameters`
- `Run` **the** `simulation`

System Specifications

System Specifications

- **Linux Mint OS version 18.3**

System Specifications

- **Linux Mint OS version 18.3**
- **OpenFOAM version 7**

System Specifications

- **Linux Mint OS version 18.3**
- **OpenFOAM version 7**
- **ParaView version 5.6.0**

System Specifications

- **Linux Mint OS version 18.3**
- **OpenFOAM version 7**
- **ParaView version 5.6.0**
- **gedit Text Editor**

Prerequisites

Prerequisites

- **You should have basic knowledge of** `turbulent flows` **and** `fluid dynamics`

Prerequisites

- **You should have basic knowledge of** turbulent flows **and** fluid dynamics
- **You should also be familiar with** simulating a flow through pipe **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

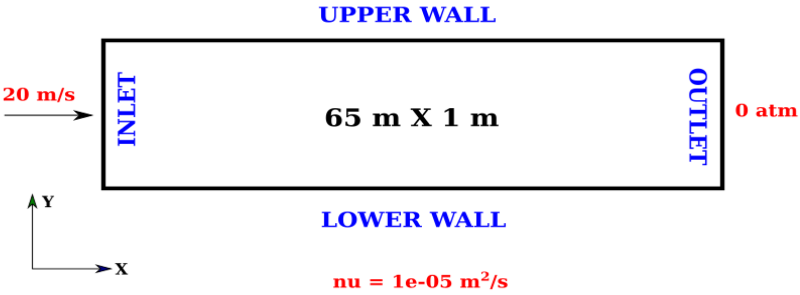
- The files used in this tutorial are available in the `Code Files` link on this tutorial page
- Please download and extract them
- Make a copy and then use them while practising

Solver detail

Solver detail

- `simpleFoam` **is a** `steady-state solver` **for** `incompressible, turbulent flow`

Problem statement



Flow properties

- Reynolds Number:

$$\mathbf{Re} = \frac{U_{avg} D}{\nu} = 2000000$$

Flow properties

- Reynolds Number:

$$\mathbf{Re} = \frac{U_{avg} D}{\nu} = 2000000$$

The flow is turbulent

K-Epsilon turbulence model

- **Widely used** `RAS turbulence model`

K-Epsilon turbulence model

- **Widely used** RAS turbulence model
- Two-equation model

K-Epsilon turbulence model

It solves

- turbulent kinetic energy transport equation

K-Epsilon turbulence model

It solves

- turbulent kinetic energy transport equation
- turbulent dissipation rate transport equation

YPlus

- Dimensionless wall distance

YPlus

- Dimensionless wall distance
- **For** wall function approach, $30 < y^+ < 300$

Wall distance, y_p

- **Distance between the** `wall` **and the nearest** `cell centre`

$$y_p = \frac{Y^+\nu}{\sqrt{0.5 C_f {U_{inf}}^2}}$$

Wall distance, y_p

- **Distance between the** `wall` **and the nearest** `cell centre`

$$y_p = \frac{Y^+\nu}{\sqrt{0.5 C_f {U_{inf}}^2}}$$

$y_p = 0.00155\ m$ **(for $y^+ = 100$)**

Wall distance, y_p

- **Distance between the** wall **and the nearest** cell centre

$$y_p = \frac{Y^+ \nu}{\sqrt{0.5 C_f U_{inf}^2}}$$

$$y_p = 0.00155\ m\ \textbf{(for } y^+ = \textbf{100)}$$

For channel flow, $C_f = 0.078 Re_{d_h}^{-1/4}$

Inlet Boundary Condition - κ

Inlet Boundary Condition - κ

turbulentIntensityKineticEnergyInlet

Inlet Boundary Condition - κ

turbulentIntensityKineticEnergyInlet

- Turbulent Intensity:
  $$I = 0.16 Re_{d_h}^{-1/8} = 0.0261$$

Inlet Boundary Condition - κ

turbulentIntensityKineticEnergyInlet

- Turbulent Intensity:
  $$I = 0.16 Re_{d_h}^{-1/8} = 0.0261$$
- Turbulent kinetic energy:
  $$\kappa = 1.5(U_{avg} I)^2 = 0.41 \ m^2/s^2$$

Inlet Boundary Condition - epsilon

Inlet Boundary Condition - epsilon

turbulentMixingLengthDissipationRateInlet

Inlet Boundary Condition - epsilon

turbulentMixingLengthDissipationRateInlet

- Turbulent length scale:
  $l = 0.07 d_h = 0.07 \ m$

Inlet Boundary Condition - epsilon

turbulentMixingLengthDissipationRateInlet

- Turbulent length scale:
  $l = 0.07 d_h = 0.07\ m$
- Turbulent dissipation:
  $\epsilon = \frac{C_\mu{}^{3/4} \kappa^{3/2}}{l} = 0.61\ m^2/s^3\ (C_\mu = 0.09)$

fossee better education

Wall Boundary Condition

Wall Boundary Condition

- Turbulent kinetic energy: kqRWallFunction

Wall Boundary Condition

- Turbulent kinetic energy: `kqRWallFunction`
- Turbulent dissipation: `epsilonWallFunction`

Outlet Boundary Condition

- Turbulent kinetic energy: zeroGradient
- Turbulent dissipation: zeroGradient

Kinematic eddy viscosity, nut

- **Dependent variable**

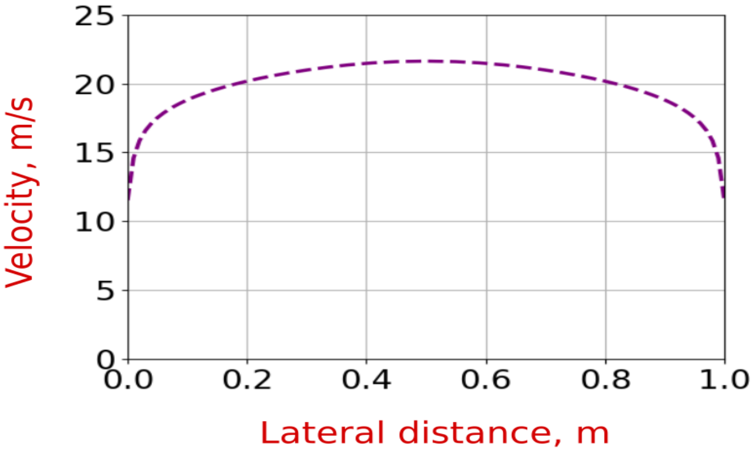
Kinematic eddy viscosity, nut

- **Dependent variable**
- **Calculated by the** `solver`

Kinematic eddy viscosity, nut

- **Dependent variable**
- **Calculated by the `solver`**
- **At `wall`, `nutWallFunction` is used**

Outlet Velocity Profile



Summary

We have learned how to:

- **Implement** k-epsilon turbulence model
- **Set up the** initial **and** boundary conditions **for** turbulence parameters
- Run **the** simulation

Assignment

- **Change the** `inlet boundary condition` **for** `kappa` **and** `epsilon` **to** `fixedValue`
- **Repeat the** `simulation` **for a** `velocity` **value of** `40 m/s`**, change** `kappa` **and** `epsilon` **accordingly**

About the Spoken Tutorial Project

- Watch the video available at 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

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**