Turbulence Modelling in OpenFOAM

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:

- **Set up the** `blockMeshDict` **dictionary for a given** `YPlus` **value**

Learning Objectives

We will learn how to:

- **Set up the** `blockMeshDict` **dictionary for a given** `YPlus` **value**
- **Implement** `k-omega` **and** `k-omega SST turbulence models`

Learning Objectives

We will learn how to:

- **Set up the** `blockMeshDict` **dictionary for a given** `YPlus` **value**
- **Implement** `k-omega` **and** `k-omega SST turbulence models`
- `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** `geometric progression`

Prerequisites

- **You should have basic knowledge of** `geometric progression`
- **You should be familiar with** `simulating a turbulent flow through a channel`

Prerequisites

- **You should also be familiar with** `Multi-block Meshing of 2D Geometry`

Prerequisites

- **You should also be familiar with** `Multi-block Meshing of 2D Geometry`
- **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

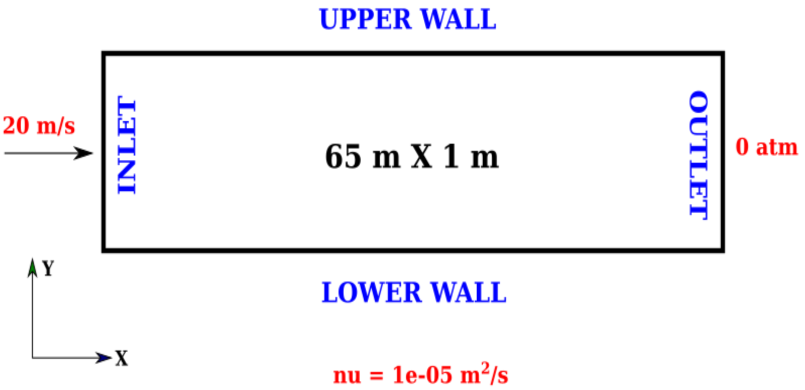
- **The files used in this tutorial are available in the** `Code Files` **link on this tutorial page**
- **Please download and extract them**
- **Make a copy and then use them while practising**

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

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`

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

$$y_p = 0.0031m \ \textbf{(for } y^+ = 200\textbf{)}$$

Wall distance, y_p

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

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

$$\color{red}{y_p = 0.0031m}\ \textbf{(for } y^+ = 200\textbf{)}$$

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

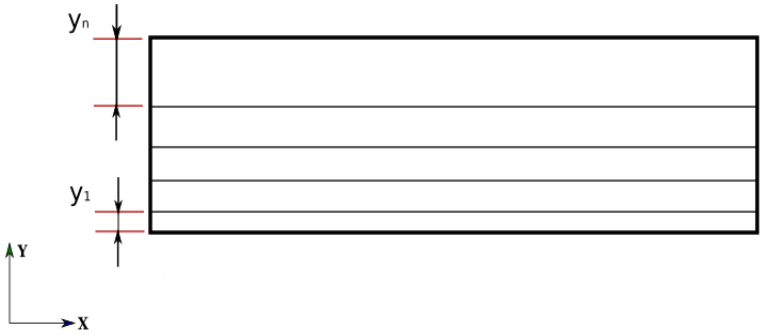
Cell width

`Width` **of the** `cell` **near** `wall`**,**

$$2 \times y_p = 0.0062$$

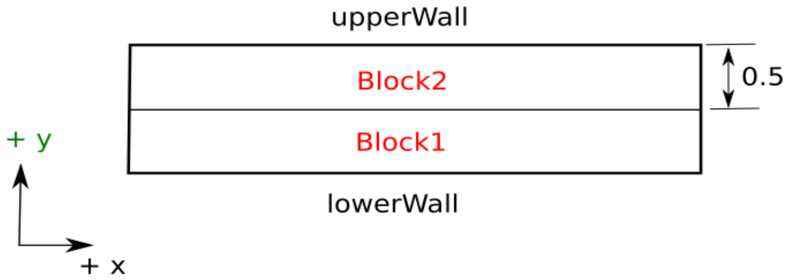
Expansion ratio

last `cell width` / **first** `cell width`



Expansion ratio calculation

Calculated using `geometric progression formula`



Expansion ratio value

The `expansion ratio` **in** `+y direction`**:**

- **For** `block 1` $= 5.57$

Expansion ratio value

The `expansion ratio` **in** `+y direction`**:**

- **For** `block 1` **= 5.57**
- **For** `block 2` **= 0.18**

Expansion ratio value

The `expansion ratio` **in** `+y direction`**:**

- **For** `block 1` = 5.57
- **For** `block 2` = 0.18

for a `block` **of** `width` = 0.5 `m`**, and** `cell number` = 30

K-Omega turbulence model

Two-equation model

K-Omega turbulence model

Two-equation model

It solves

- turbulent kinetic energy transport equation

K-Omega turbulence model

Two-equation model

It solves

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

Inlet Boundary Condition - omega

Inlet Boundary Condition - omega

turbulentMixingLengthFrequencyInlet

Inlet Boundary Condition - omega

turbulentMixingLengthFrequencyInlet

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

Inlet Boundary Condition - omega

- Specific turbulent dissipation rate:

$$\omega = C_{\mu}^{-1/4}\frac{\sqrt{\kappa}}{l} = 16.67\ 1/s$$

where,

empirical constant, $C_{\mu} = 0.09$

Wall and outlet Boundary Condition - omega

Wall:

`omegaWallFunction`

Wall and outlet Boundary Condition - omega

Wall:

`omegaWallFunction`

Outlet:

`zeroGradient`

K-Omega SST turbulence model

Hybrid model:

K-Omega SST turbulence model

Hybrid model:

- Near-wall **region**

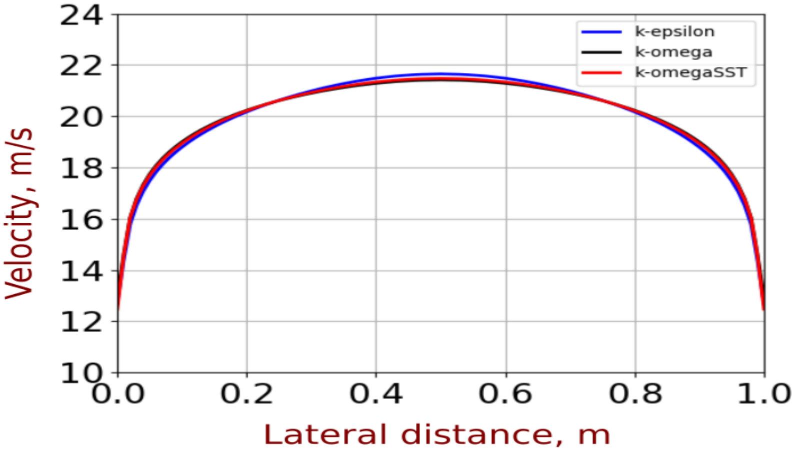
  k-omega

K-Omega SST turbulence model

Hybrid model:

- Near-wall **region**
  k-omega
- **Region away from** wall
  k-epsilon

Exit velocity profile



Summary

We have learnt to:

- **Set up the** `blockMeshDict` **file for a given** `YPlus` **value**
- **Implement** `k-omega` **and** `k-omega SST turbulence models`
- `Run` **the** `simulation`

Assignment

- **Repeat the** `simulation` **for a** `YPlus` **value of** `250` **using** `k-epsilon model`
- **Repeat the** `simulation` **for a** `velocity` **of** `40 m/s` **using** `k-omega` **and** `k-omega SST model`

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**