Creating 3D Pipe Geometry and Mesh in OpenFOAM

Spoken Tutorial Project
https://spoken-tutorial.org
National Mission on Education through ICT
http://sakshat.ac.in/
Divyesh Variya & Swetha Sridhar
IIT Bombay

June 19, 2020

Learning Objectives

We will learn how to:

Learning Objectives

We will learn how to:

- **Create a** `3D Geometry` **using** `blockMeshDict`

Learning Objectives

We will learn how to:

- **Create a** `3D Geometry` **using** `blockMeshDict`
- `Mesh` **a** `3D Geometry`

Learning Objectives

We will learn how to:

- **Create a** `3D Geometry` **using** `blockMeshDict`
- `Mesh` **a** `3D Geometry`
- **Label the** `boundary patches`

Learning Objectives

- Check **the** mesh results **using** checkMesh **command**

Learning Objectives

- Check the mesh results using checkMesh command
- View the 3D Geometry and Mesh in ParaView

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

- **You should be familiar with creation of a basic geometry using the `blockMesh` utility**

Prerequisites

- **You should be familiar with creation of a basic geometry using the `blockMesh` utility**
- **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

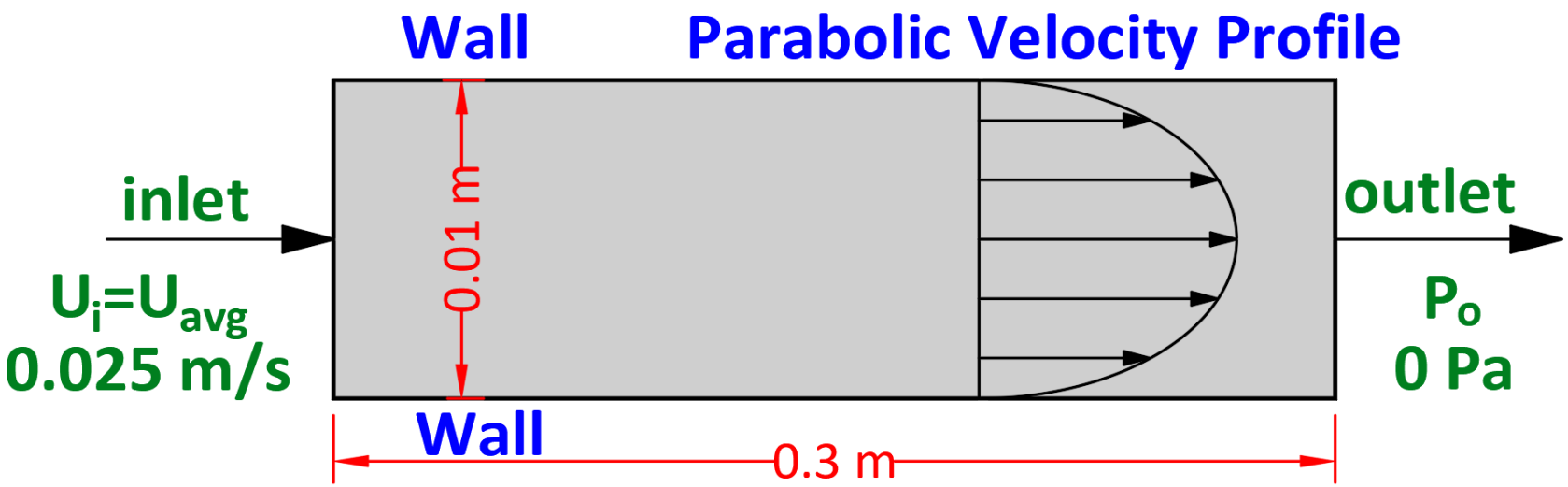
- **The files used in this tutorial are available in the `Code Files` link on this tutorial page**
- **Please download and extract them**

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**

Hagen Poiseuille flow through a pipe

Hagen Poiseuille flow through a pipe

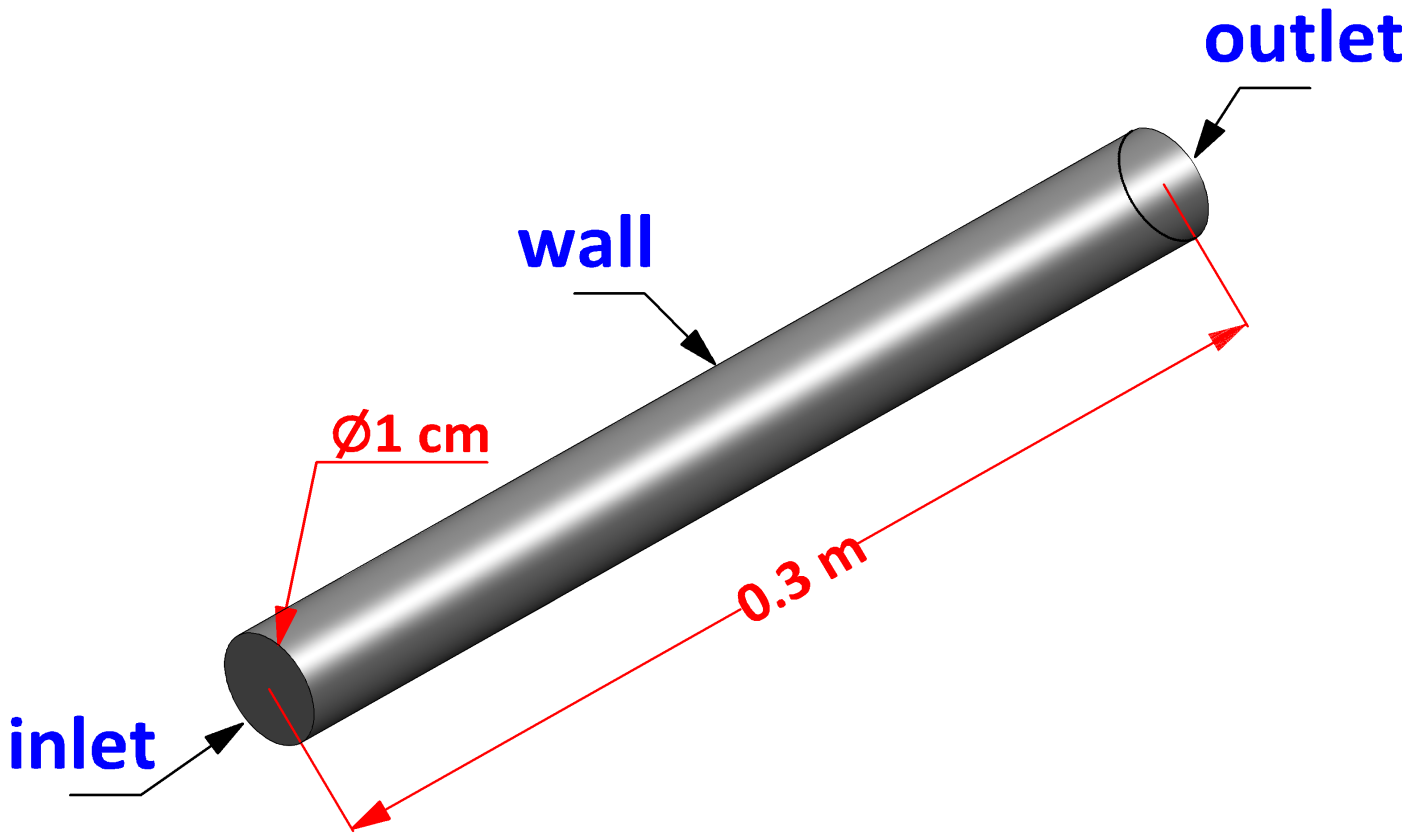


For water,

μ = Dynamic Viscosity = 1e-03

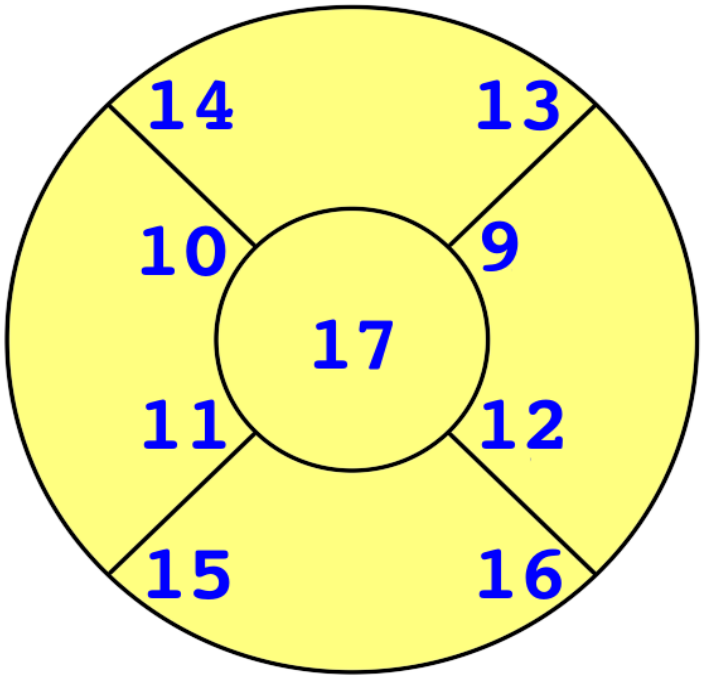
η = Kinematic Viscosity = 1e-06

Pipe Geometry



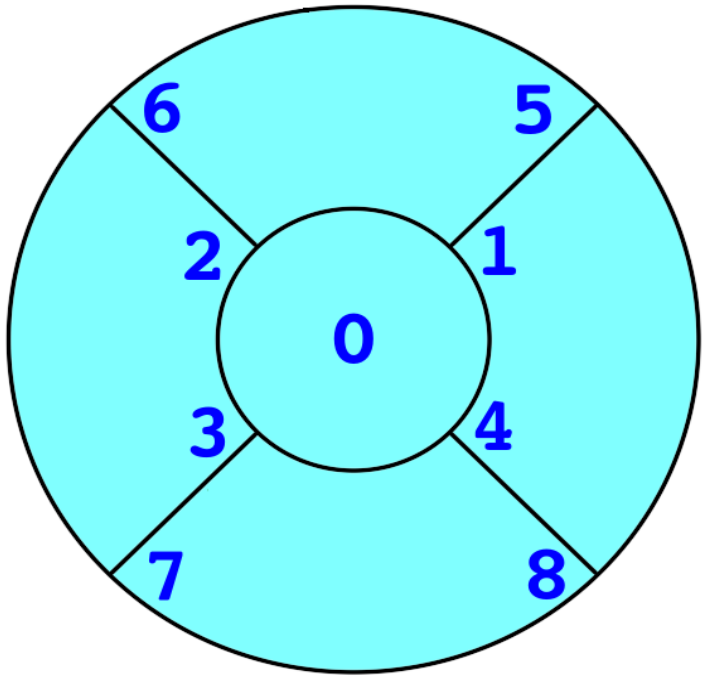
Vertices Detail

Front Face



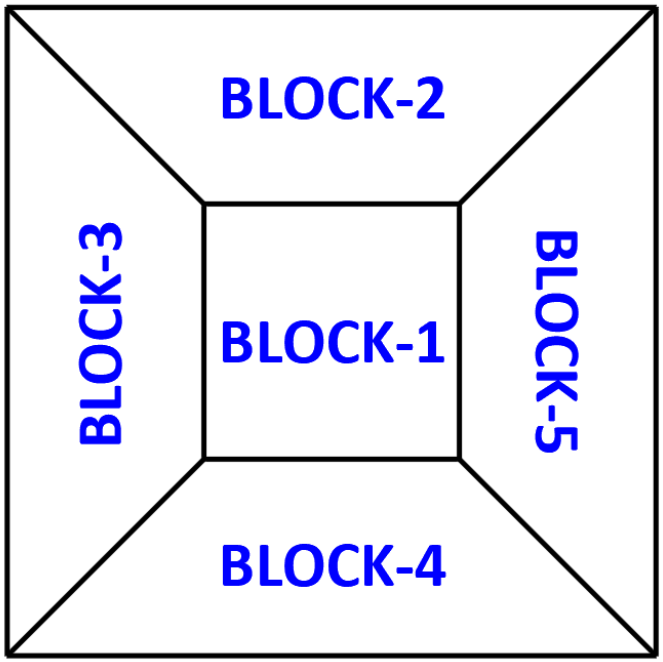
(outlet)

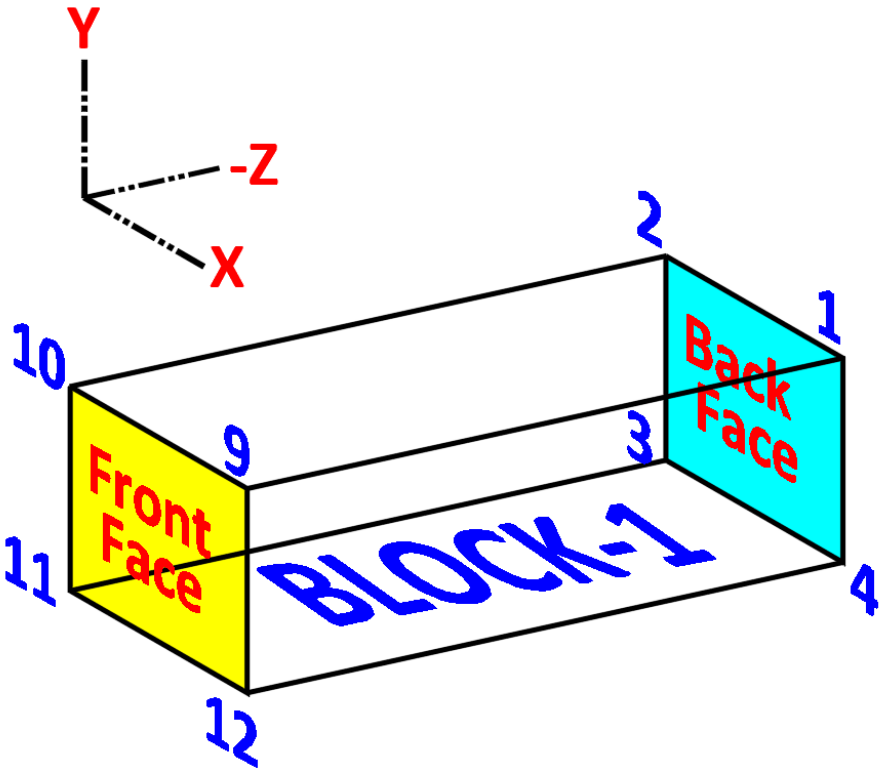
Back Face



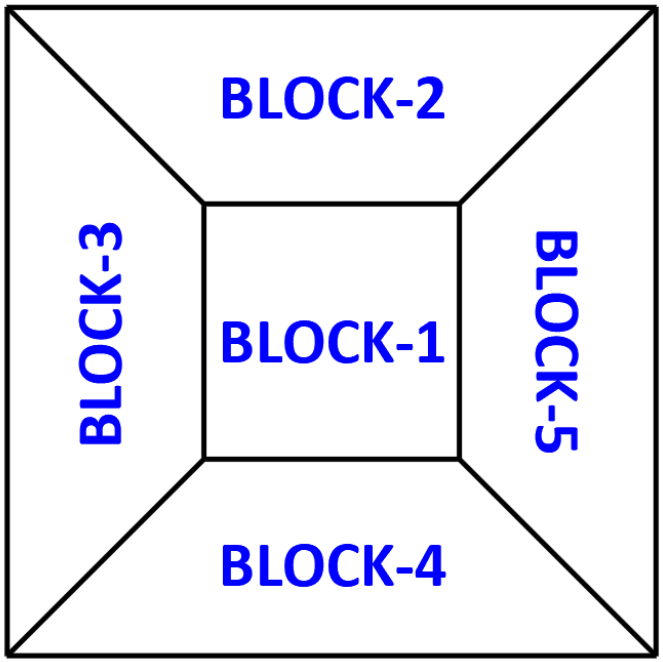
(inlet)

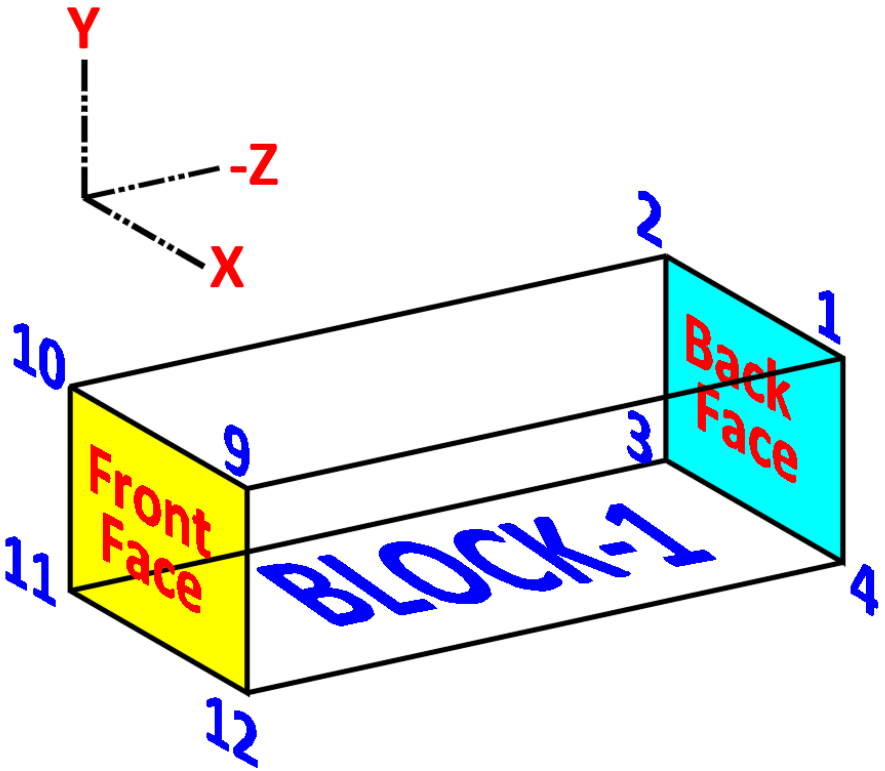
Block Detail





Block Detail





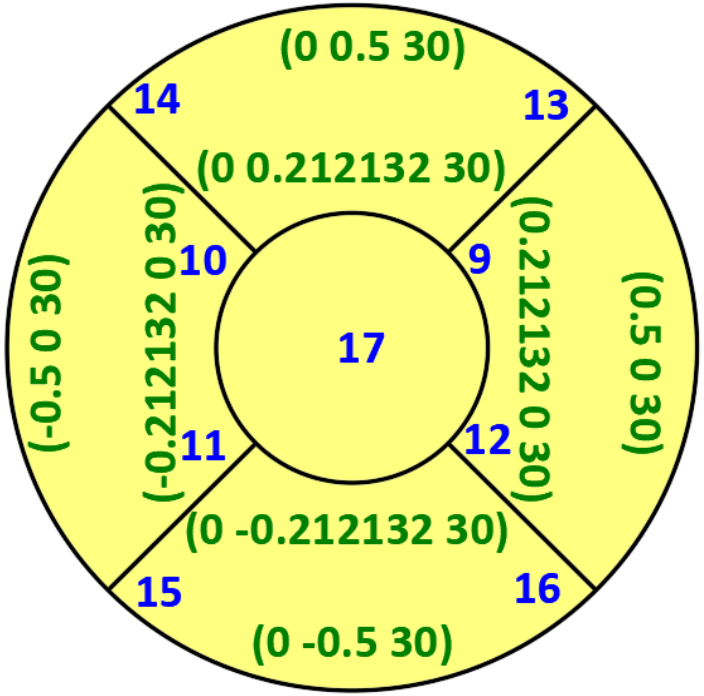
hex (1 2 3 4 9 10 11 12) (8 8 80)

simpleGrading (1 1 1)

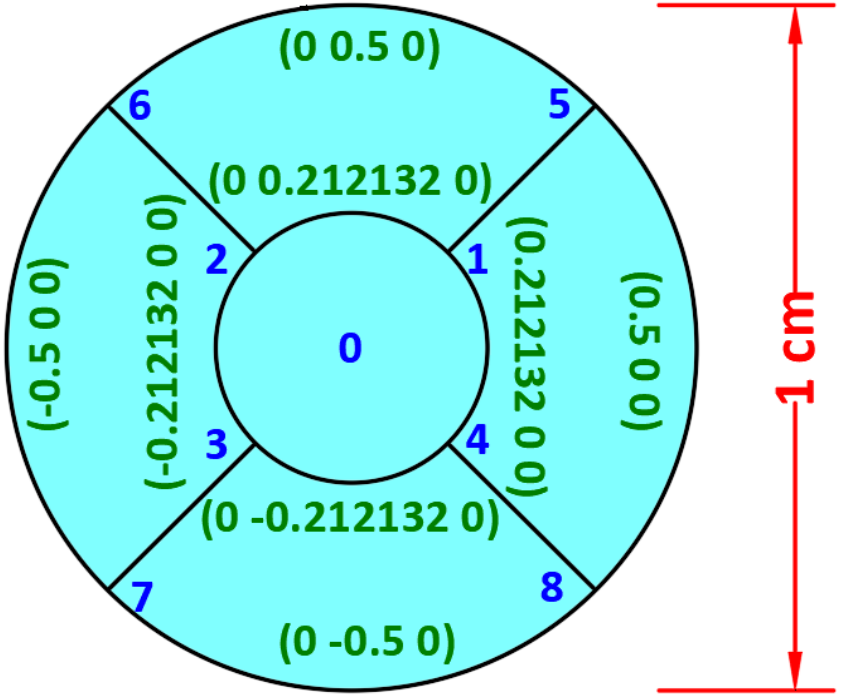
Geometry Arc Detail

Geometry Arc Detail

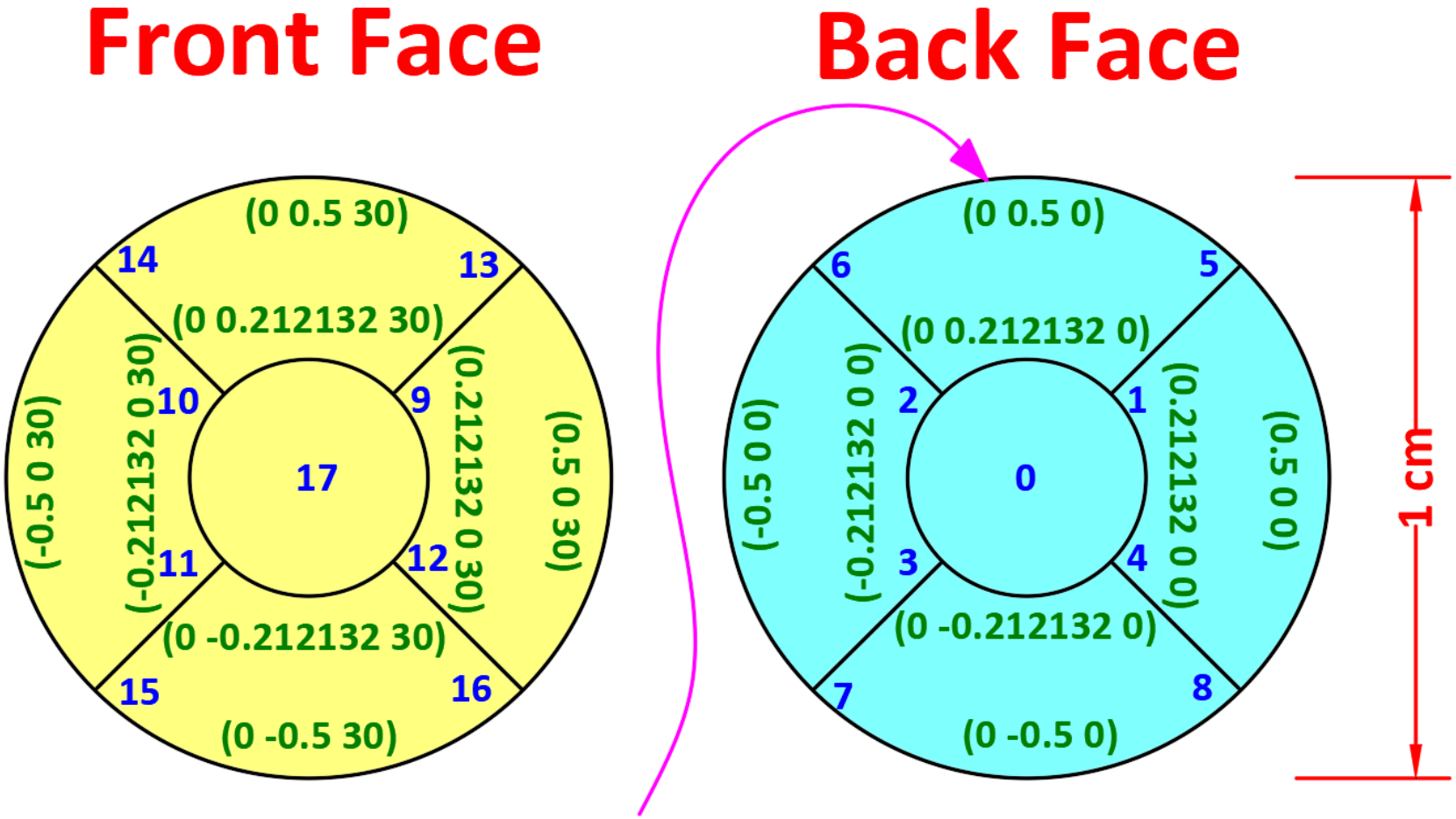
Front Face



Back Face



Geometry Arc Detail



Summary

We have learnt how to:

- **Create a** `3D Geometry` **using** `blockMeshDict`
- **Mesh a** `3D Geometry`
- **Label the** `boundary patches`

Summary

- Check the mesh results using checkMesh command
- View the 3D Geometry and Mesh in ParaView

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**