

Simulating 1D Conduction through a Bar

Spoken Tutorial Project
<https://spoken-tutorial.org>

National Mission on Education through ICT

Mano Prithvi Raj, Aabhushan Regmi,
Payel Mukherjee

IIT Bombay

5 June 2023



Learning Objectives

We will learn to:



Learning Objectives

We will learn to:

- ▶ **Set up a case of** heat transfer
in OpenFOAM



Learning Objectives

We will learn to:

- ▶ **Set up a case of** heat transfer in OpenFOAM
- ▶ **Simulate a** conduction heat transfer case **using a** laplacianFoam **solver**



System Specifications



System Specifications

► Ubuntu Linux OS version 22.04



System Specifications

- ▶ **Ubuntu Linux OS version 22.04**
- ▶ **OpenFOAM version 9**



System Specifications

- ▶ **Ubuntu Linux OS version 22.04**
- ▶ **OpenFOAM version 9**
- ▶ **ParaView version 5.6.0**



System Specifications

- ▶ **Ubuntu Linux OS version 22.04**
- ▶ **OpenFOAM version 9**
- ▶ **ParaView version 5.6.0**
- ▶ **gedit Text Editor**



Prerequisites



Prerequisites

- ▶ **You should have basic knowledge of conductive heat transfer**



Prerequisites

- ▶ **You should have basic knowledge of** conductive heat transfer
- ▶ **You should be familiar with** setting up a case 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 provided in the Code Files link on this tutorial page**



Code Files

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

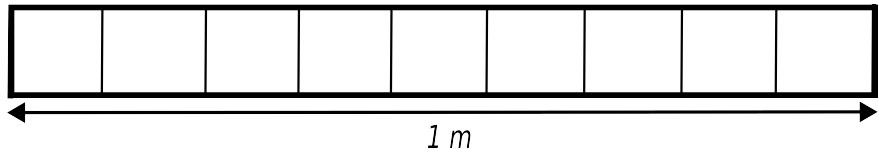


Code Files

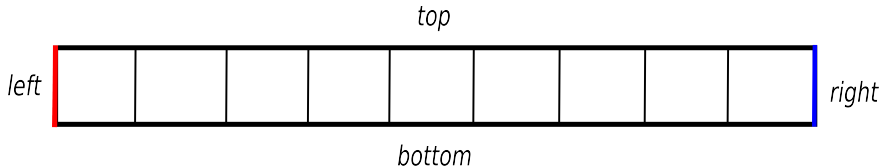
- ▶ The files used in this tutorial are provided in the `Code Files` link on this tutorial page
- ▶ Please download and extract them
- ▶ Make a copy and then use them while practising



Geometry



Geometry



laplacianFoam



laplacianFoam

- ▶ `laplacianFoam` **is a basic**
`OpenFOAM` **solver**



laplacianFoam

- ▶ `laplacianFoam` **is a basic** OpenFOAM **solver**
- ▶ `laplacianFoam` **solves simple Laplace equations**



laplacianFoam

- ▶ `laplacianFoam` **is a basic OpenFOAM solver**
- ▶ `laplacianFoam` **solves simple Laplace equations**
- ▶ **An example of such an equation is thermal diffusion in a solid**



laplacianFoam



laplacianFoam

- ▶ **This is the equation implemented in `laplacianFoam`**



laplacianFoam

- ▶ **This is the equation implemented in `laplacianFoam`**

$$\frac{\partial T}{\partial t} = \nabla \cdot (\alpha \nabla T), \quad (1)$$

where

- ▶ **α is the thermal diffusivity**



laplacianFoam

- ▶ **This is the equation implemented in `laplacianFoam`**

$$\frac{\partial T}{\partial t} = \nabla \cdot (\alpha \nabla T), \quad (1)$$

where

- ▶ **α is the thermal diffusivity**
- ▶ **T is temperature**



Boundary Conditions



$$T_{left} > T_{right}$$



Boundary Conditions

Boundary	Temperature
left	373 K
right	273 K
topAndBottom	empty
frontAndBack	empty



Summary

We have learnt to:

- ▶ **View how laplacian equation is implemented in OpenFoam**
- ▶ **Solve heat transfer problem using OpenFoam**
- ▶ **Post-process results in paraview**



Assignment

- ▶ **Increase the length of the bar in the x-direction to**
2 m
- ▶ **Change the DT value to 0.005**
- ▶ **Keep all the other parameters unaltered in your simulation**



Assignment

- ▶ Simulate **the** conduction heat transfer **through this** bar
- ▶ **View the** temperature contours



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 was established by the Ministry of Education, Government of India**

