

Installing OpenFOAM in Ubuntu

Spoken Tutorial Project

<https://spoken-tutorial.org>

National Mission on Education through ICT

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

Ashley Melvin & Padmini Priyadarshini

IIT Bombay

March 13, 2020



Learning Objectives

We will learn how to:



Learning Objectives

We will learn how to:

- ▶ **Install OpenFOAM version 7**



Learning Objectives

We will learn how to:

- ▶ **Install OpenFOAM version 7**
- ▶ **Install ParaView version 5.6.0**



System Specifications



System Specifications

► Ubuntu Linux OS version 18.04



System Specifications

- ▶ **Ubuntu Linux OS version 18.04**
- ▶ **A working Internet connection**



System Specifications

- ▶ **Ubuntu Linux OS version 18.04**
- ▶ **A working Internet connection**
- ▶ **A text editor**



Summary

We have learnt how to:

- ▶ **Install OpenFOAM version 7**
- ▶ **Install ParaView version 5.6.0**
- ▶ **Verify the installation**



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**

