Basic Post-Processing using ParaView

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

Script : Subhasree Basu
Narration : Deepa Vedartham

IIT Bombay

24th July, 2017

Learning Objectives

In this tutorial, we will learn:

- **Some basic visualization techniques in ParaView**

Learning Objectives

Under this we will learn:

- **Variable Visualization**

Learning Objectives

Under this we will learn:

- **Variable Visualization**
- **Velocity vector Visualization**

Learning Objectives

Under this we will learn:

- **Variable Visualization**
- **Velocity vector Visualization**
- **Streamlines Visualization**

Learning Objectives

Under this we will learn:

- **Variable Visualization**
- **Velocity vector Visualization**
- **Streamlines Visualization**
- **Slice and Clip filters**

System Requirement

- **Ubuntu Linux Operating system 16.04 LTS**

System Requirement

- **Ubuntu Linux Operating system 16.04 LTS**
- **OpenFOAM version 4.1**

System Requirement

- **Ubuntu Linux Operating system 16.04 LTS**
- **OpenFOAM version 4.1**
- **ParaView version 5.0**

Pre-requisite

- **Basic knowledge of Linux terminal commands**

Pre-requisite

- **Basic knowledge of Linux terminal commands**
- **Experience with simulating cases in OpenFOAM**

Pre-requisite

- **Basic knowledge of Linux terminal commands**
- **Experience with simulating cases in OpenFOAM**
- **Working skills in ParaView**

Pre-requisite

If not, please go through the previous OpenFOAM Spoken Tutorials on this website:

http://spoken-tutorial.org/

Problem Statement-Variable Visualization

- **OpenFOAM has provided an inbuilt case file for pitzDaily**

Problem Statement-Variable Visualization

Here we will:

Problem Statement-Variable Visualization

Here we will:

- **focus on post-processing of the pitzDaily solution**

Problem Statement-Variable Visualization

Here we will:

- **focus on post-processing of the pitzDaily solution**
- **learn to visualize flow field variables like velocity, pressure, vorticity, etc**

Problem Statement-Variable Visualization

We will also:

Problem Statement-Variable Visualization

We will also:

- **learn to visualize velocity vector within the flow field**

Code File

You can find the `pitzDaily case` **file within the** `simpleFoam` **directory within** `OpenFOAM-4.0>tutorials` **directory**

Problem Statement-Streamlines Visualization

- **OpenFOAM has provided an inbuilt case file for flow over a motorBike**

Problem Statement-Streamlines Visualization

Here we will:

- **focus on post-processing of the motorBike solution**

Problem Statement-Streamlines Visualization

Here we will:

- **focus on post-processing of the motorBike solution**
- **learn to create stream lines and view them in ParaView**

Code File

You can find the `motorBike case` **file within the** `simpleFoam` **directory within** `OpenFOAM-4.0>tutorials` **directory**

Problem Statement-Clip and Slice Filters

- **OpenFOAM has provided an inbuilt case file for hotRadiationRoom**

Problem Statement-Clip and Slice Filters

Here we will:

- **focus on post-processing of the hotRadiationRoom solution**

Problem Statement-Clip and Slice Filters

Here we will:

- **focus on post-processing of the hotRadiationRoom solution**
- **learn to use filters like clip and slice in ParaView**

Code File

- **To open the** `hotRadiationRoom case` **file**
- **Go to the** `hotRadiationRoom` **tutorial in** `BuoyantSimpleFoam` **within** `heatTransfer case` **directory**
- **You can find this within** `OpenFOAM-4.1>tutorials` **directory**

Summary

In this tutorial, we learnt some Basic Post-Processing using ParaView

Summary

Under this we have learnt to:

- **Visualize field variable**

Summary

Under this we have learnt to:

- **Visualize field variable**
- **Create Velocity Vectors**

Summary

Under this we have learnt to:

- **Visualize field variable**
- **Create Velocity Vectors**
- **Create Stream lines**

Summary

Under this we have learnt to:

- **Visualize field variable**
- **Create Velocity Vectors**
- **Create Stream lines**
- **Use clip and slice filters**

Exercise

- **You can use the discussed utilities on other inbuilt case files**

Exercise

- **You can use the discussed utilities on other inbuilt case files**
- **This will help you understand the post-processing using ParaView better**

Forum to answer questions

- **Do you have questions on THIS Spoken Tutorial?**
- **Choose the minute and second where you have the question.**
- **Explain your question briefly.**
- **Someone from the FOSSEE team will answer them. Please visit**

http://forums.spoken-tutorial.org/

Forum to answer questions

- Questions not related to the Spoken Tutorial?
- Do you have general / technical questions on the Software?
- Please visit the FOSSEE Forum
  http://forums.fossee.in/
- Choose the Software and post your question.

Case Study Project

- **The FOSSEE team coordinates solving past, current or new CFD projects using OpenFOAM**
- **We give honorarium and certificate to those who do this**

For more details, please visit this site:

http://cfd.fossee.in/

Acknowledgements

- **Spoken Tutorial Project is a part of the Talk to a Teacher project**
- **It is supported by the National Mission on Education through ICT, MHRD, Government of India**
- **More information on this Mission is available at:**

http://spoken-tutorial.org/NMEICT-Intro

THANK YOU!

For more information, visit our website http://fossee.in/