Exporting the Geometry from Salome to OpenFOAM

Talk to a Teacher

National Mission on Education through ICT

Saurabh Sawant

Date: June 17, 2013

Talk to a Teacher

Learning Objectives

- **Group the meshed geometry parts in Salome**

Learning Objectives

- **Group the meshed geometry parts in Salome**

Learning Objectives

- **Group the meshed geometry parts in Salome**
- **Export the geometry to OpenFOAM**

Talk to a Teacher

Learning Objectives

- **Group the meshed geometry parts in Salome**

- **Export the geometry to OpenFOAM**

Learning Objectives

- **Group the meshed geometry parts in Salome**

- **Export the geometry to OpenFOAM**

- **Create a case directory for simulation**

Learning Objectives

- **Group the meshed geometry parts in Salome**
- **Export the geometry to OpenFOAM**
- **Create a case directory for simulation**

Learning Objectives

- **Group the meshed geometry parts in Salome**

- **Export the geometry to OpenFOAM**

- **Create a case directory for simulation**

- **View geometry in ParaView**

System Requirement

- **Linux Operating System Ubuntu version 12.10**

System Requirement

- **Linux Operating System Ubuntu version 12.10**

System Requirement

- **Linux Operating System Ubuntu version 12.10**
- **OpenFOAM version 2.1.1**

System Requirement

- **Linux Operating System Ubuntu version 12.10**
- **OpenFOAM version 2.1.1**

System Requirement

- **Linux Operating System Ubuntu version 12.10**
- **OpenFOAM version 2.1.1**
- **ParaView version 3.12.0**

Talk to a Teacher

System Requirement

- **Linux Operating System Ubuntu version 12.10**
- **OpenFOAM version 2.1.1**
- **ParaView version 3.12.0**

Talk to a Teacher

System Requirement

- **Linux Operating System Ubuntu version 12.10**
- **OpenFOAM version 2.1.1**
- **ParaView version 3.12.0**
- **Salome version 6.6.0**

Talk to a Teacher

- **Creating and meshing a Curved-Pipe Geometry in Salome**

Summary

- **Group the meshed geometry parts in Salome**

Summary

- **Group the meshed geometry parts in Salome**
- **Export the geometry to OpenFOAM**

Summary

- **Group the meshed geometry parts in Salome**
- **Export the geometry to OpenFOAM**
- **Create a case directory for simulation**

Summary

- **Group the meshed geometry parts in Salome**
- **Export the geometry to OpenFOAM**
- **Create a case directory for simulation**
- **View geometry in ParaView**

Assignment

- **Run the simulation by making appropriate changes in the files as described**

- **Run the simulation by making appropriate changes in the files as described**

- **Export the geometries that you have created on your own**

Talk to a Teacher

Assignment

- **Run the simulation by making appropriate changes in the files as described**

- **Export the geometries that you have created on your own**

- **Run the simulations on those geometries**

About the Spoken Tutorial Project

- Watch the video available at http://spoken-tutorial.org /What_is_a_Spoken_Tutorial
- It summarises the Spoken Tutorial project

Talk to a Teacher

About the Spoken Tutorial Project

- Watch the video available at http://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

Talk to a Teacher

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**

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**