Introduction to OpenFOAM & paraView

Course Syllabus

Instructor:
Dr. Siva Parameswaran
Professor, Department of Mechanical Engineering
Email: siva.parameswaran@ttu.edu

Course Information:

- **Reference Books:**
  - *a)* *Modern Compressible Flow: With Historical Perspective*, John D. Anderson.
  - *b)* *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, H Versteeg & W Malalasekera.
  - *c)* *Numerical Heat Transfer and Fluid Flow: Computational Methods in Mechanics & Thermal Sciences*, Suhas V. Patankar.
  - *d)* *Computational Methods for Fluid Dynamics*, Joel H. Ferziger, Milovan Peric.
  - *e)* *A Friendly Introduction to Numerical Analysis*, Brian Bradie.
  - *f)* *Advanced Engineering Mathematics*, Erwin Kreyszig.
- **Softwares:**
  - a) VirtualBox (Ubuntu OS)
  - b) OpenFOAM & paraview (7 & above)

  Note: (All these are Free & opensource s/w)
- **Description:** Introduction and Installation; CFD overview with Linux basics, Meshing in OpenFOAM, Types of Errors and Debugging; Next Steps to Master OpenFOAM.
- **Purpose:** This course presents an overview of Computational Fluid Dynamics and gives a working idea about **OpenFOAM** and **paraview**. Learning the workflow to setup a CFD simulation in OpenFOAM helps understand the working process of popular commercial CFD codes like Fluent, Solid works Flow simulation, Acusolve, Converge…etc.
- **Prerequisites:** Basic Fluid Mechanics Concepts & Vector Calculus.
- **Laptop**: Minimum 4GB RAM PC with decent internet connection.
- End of Course Feedback Form to be filled after completing the Course.

Grading Methods:

- **Quiz:**
  Students will be given a set of questions after every lesson to test their knowledge on the given section of the course. If the student procures 80% and above in that section, he/she will be allowed to move on to the next section. On the other hand, if he/she fails to score above 80% must retake the test to proceed to the next lesson.
- **Projects:**
  This course has two projects **(optional).** They involve submission of a detailed report on the investigation asked to do. Though this is optional, students who complete the same, will have an opportunity to be featured on the website.

  Note: Queries on any content posted on the website need to be addressed to support@katrinsakthi.com

Expected Learning Outcomes / Assessment Methods:

Expected Learning Outcome	Assessment Methods
• Basic Linux Commands	Quiz
• CFD workflow & OpenFOAM code structure	Quiz
• Meshing in OpenFOAM	Quiz
• Common Errors in OpenFOAM simulation	Quiz
• Post-process using paraview	Quiz
• Setting up a CFD simulation and running a tutorial case in OpenFOAM	Projects

Schedule (subjected to change):

Lesson	Video #	Duration	Topics
1	1	0:06:00	Introduction to OpenFOAM & paraview
	2	0:13:43	Installation Help
2	3	0:16:15	Basic Linux Commands
	4	0:06:31	Important CFD terms to Know
	5	0:08:51	CFD Workflow and structure of OpenFOAM
3	6	0:09:52	BlockMesh in OpenFOAM Part 1
	7	0:09:29	BlockMesh in OpenFOAM Part 2
	8	0:10:29	OpenFOAM Simulation Walkthrough
	9	0:16:26	Governing equations & Solvers in OF
4	10	0:04:22	Common Simulation Errors
	11	0:09:41	Post-processing using paraview
	12	-	Final Project 1
5	13	-	Final Project 2(optional)
	14	-	Feedback Form & Next Steps with summary
	15	-	Promo to next course

ACADEMIC INTEGRITY:
Plagiarism of any sort will be seriously viewed upon.