Applied Computational Fluid Dynamics Using OpenFOAM

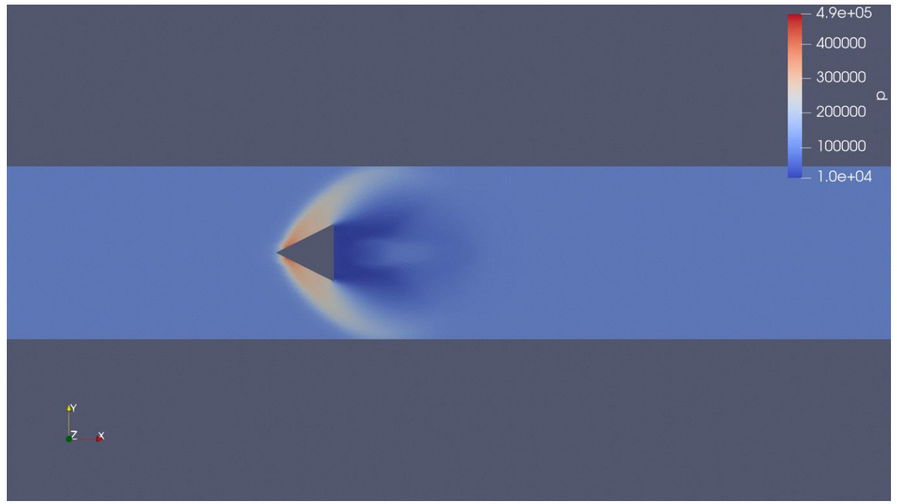
Value Added Course
College/University: KSR
Spring 2025

ExaSlate
Develop = Guide = Collab

ExaSlate
Develop = Guide = Collab

Overview

- Brief Introduction
- Installations



Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Brief Introduction

- Instructors:
  - Lakshman Anumolu, Kumaresh Selvakumar, Shyam Sundar J
- Course duration per session: 45 mins – 1.5 hrs
- Requirements:
  - Virtual box and installing OS & softwares.
  - Interest to learn CFD using OpenFOAM & Octave
  - **Interest to ask questions**
  - **Work as a team**
- Exercises/Projects: equal weightage
- Final grade/performance is used as one of the criterion for internship and other opportunities

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Important Links

- Course repository
  - https://github.com/exaslate-learn/applied-cfd-using-openfoam-ksr-spring-2025

- WhatsApp
  - https://chat.whatsapp.com/IfbuHs6P4cu6iLxvdyr8Zf

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

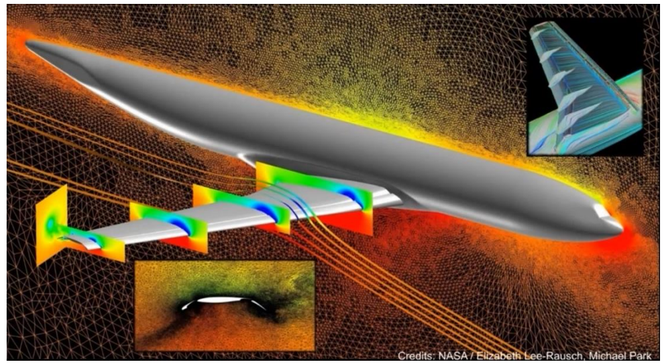
Objective

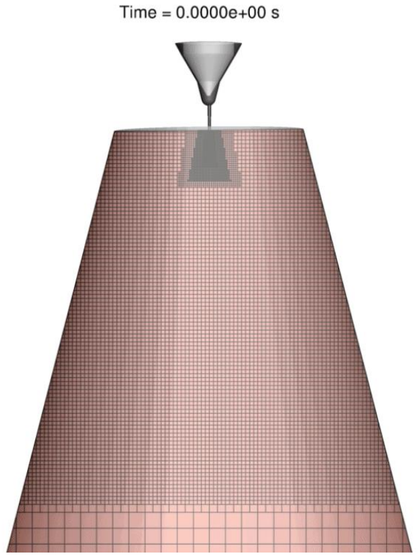
- Minimal Theory
- Simulate what you have studied in textbooks
- Some practical examples that could be useful for your projects

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

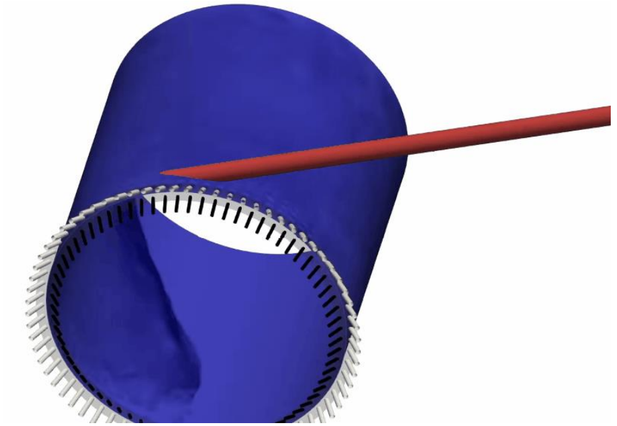
Objective





Why CFD?

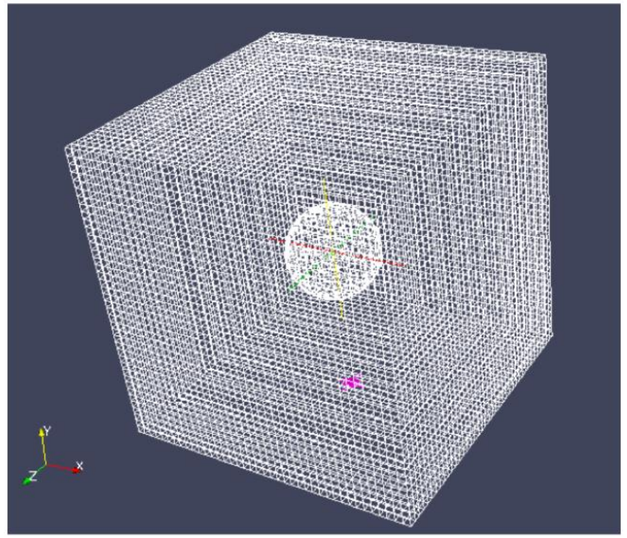
- Speed of performing tests for different scenarios
- Cost
- Parametric study



Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

CFD Workflow



Generate mesh

OpenFOAM simulation

Perform simulation

Post-process results

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Tools for this Course

- Operating System:
  - Ubuntu 24.04

Ubuntu 24.04 LTS

Exercise-1

Install required tools

- Softwares:
  - OpenFOAM v2412

OpenFOAM®

OpenFOAM-v2412 openfoam

  - Octave

GNU Octave

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Things to do by You

- Create a github account:
  - https://github.com
  - Discussion forum:
    - https://github.com/exaslate-learn/applied-cfd-using-openfoam-ksr-spring-2025/discussions
- Queries
  - Ask in github discussion forums or WhatsApp group
- Exercises/Projects
  - First exercise: https://github.com/exaslate-learn/applied-cfd-using-openfoam-ksr-spring-2025/discussions/2

Kumaresh Selvakumar
kumaresh@exaslate.com

THANK YOU