

Week 1 Progress Report

Anugya Chaubey

Semester-Long Internship 2026
OpenFOAM GUI

March 14, 2026

Project Overview

Project Objective

The goal of this project is to extend the **CFDof add-on in FreeCAD** so that more **OpenFOAM simulation features can be accessed through a graphical interface**.

Currently, many advanced OpenFOAM features require manual editing of configuration files, which makes the workflow difficult for beginners.

Expected Improvements

- ◀ More simulation options available in the FreeCAD interface
- ◀ Reduced need to manually edit OpenFOAM files
- ◀ Improved usability for beginners working with CFD simulations

Topics Studied in Week 1

During Week 1, the focus was on understanding the basic tools and concepts required for the project.

Topics Studied

- ◀ Introduction to Computational Fluid Dynamics (CFD)
- ◀ Overview of FreeCAD software and its architecture
- ◀ Understanding FreeCAD workbenches
- ◀ Basics of 3D geometry creation in FreeCAD
- ◀ Introduction to the CFD workflow used in engineering simulations

These topics helped build the foundation needed to work with **CFDof** and **OpenFOAM integration** in later stages of the project.

FreeCAD Exploration

In this week, I explored the FreeCAD interface and basic modeling tools.

Activities Performed

- ◀ Installed FreeCAD and explored its interface
- ◀ Learned about different workbenches such as **Part** and **Part Design**
- ◀ Created simple 3D models including:
 - Cube
 - Cylinder
 - Pipe (hollow cylinder using Boolean operations)

Key Learning

FreeCAD uses **parametric modeling**, which means the dimensions of objects can be modified later and the model automatically updates.

Understanding the CFD Workflow

Typical CFD Workflow

- 1 Geometry Creation A 3D model of the system is created using CAD software.
- 2 Mesh Generation The geometry is divided into small cells called mesh elements.
- 3 Boundary Conditions Conditions such as inlet velocity, outlet pressure, and wall behavior are defined.
- 4 Solver Execution Numerical equations are solved to simulate fluid flow.
- 5 Post-Processing Results such as velocity and pressure fields are visualized.

Understanding this workflow is important because **FreeCAD** and **OpenFOAM** work together to perform these steps.

Key Learnings from Week 1

The first week helped build a strong foundation for the project.

Major Learnings

- ◀ Basic understanding of Computational Fluid Dynamics
- ◀ Familiarity with FreeCAD interface and modeling tools
- ◀ Ability to create basic geometries required for simulations
- ◀ Understanding how CAD models are used in CFD simulations
- ◀ Initial understanding of how FreeCAD can be connected with simulation tools like OpenFOAM

These concepts will be important for implementing **advanced simulation features in the CFDof add-on.**