

Week 7 Progress Report

Anugya Chaubey

Semester-Long Internship 2026
OpenFOAM GUI

April 26, 2026

Project Overview

OpenFOAM CFD Case Setup

- ◀ FreeCAD CfdOF generates OpenFOAM cases
- ◀ User defines geometry, physics, turbulence
- ◀ Write Case creates OpenFOAM files
- ◀ Script automates fixes and execution

Pipeline

FreeCAD → Write Case → Script → OpenFOAM Solver

System Configuration

Parameter	Value
OpenFOAM Version	2306
Solver	pimpleFoam
Mesh Generator	cfMesh
Parallel Execution	4 cores (MPI)
OS	WSL2 Ubuntu
Automation Script	runCFD.sh

Key Problems Identified

- ◀ Missing pressure reference (pRefCell)
- ◀ momentumTransport file not generated
- ◀ Invalid include in controlDict
- ◀ Slow solver performance
- ◀ Solver block mismatch

Goal: Fix OpenFOAM-side issues without modifying user physics

Error 1: pRefCell Missing

Error

```
FOAM FATAL IO ERROR:  
Please supply pRefCell or pRefPoint
```

Fix

```
SIMPLE  
{  
    pRefCell 0;  
    pRefValue 0;  
}
```

Reason

- ◀ pimpleFoam still checks SIMPLE block

Error 2: momentumTransport Missing

Problem

- ◀ OpenFOAM 2306 uses momentumTransport
- ◀ CfdOF generates turbulenceProperties only

Fix

- ◀ Copy turbulenceProperties
- ◀ Rename object field

Key Idea

No physics modified — only structure fixed

Error 3: Invalid Include

Problem

```
#include "turbulenceLib"
```

Fix

```
sed -i '/turbulenceLib/d' system/controlDict
```

Performance Optimization

Optimization	Speedup
Binary mesh format	3–5x
Parallel (4 cores)	3–4x
Reduced mesh size	5–10x
Reduced write frequency	1.5x
Total	20–50x

Solver Configuration Insight

- ◀ pimpleFoam uses PIMPLE block
- ◀ SIMPLE block required for pressure reference

Important

Do NOT replace PIMPLE with SIMPLE

Automation Script Overview

runCFD.sh

- ◀ Handles all fixes automatically
- ◀ Runs meshing + solver
- ◀ Converts formats and cleans files

Automation Workflow



Key Design Principle

Nothing is hardcoded

- ◀ User defines physics in FreeCAD
- ◀ Script does only structural fixes
- ◀ No turbulence values modified

User Workflow

- 1 Modify geometry / physics in FreeCAD
- 2 Click Write Case
- 3 Run automation script

Result: Fully automated simulation

Mesh Recommendations

- ◀ Base size: 10–20% of geometry
- ◀ Target cells: 100k–300k
- ◀ Refinement: minimal
- ◀ Enable binary mesh

File Structure

File	Location	Purpose
controlDict	system/	Time settings
fvSolution	system/	Solver config
turbulenceProperties	constant/	User physics
momentumTransport	constant/	Solver config
runCFD.sh	Temp	Automation

Final Outcome

- ◀ All errors resolved
- ◀ Automation fully working
- ◀ Solver runs successfully
- ◀ No manual fixes required

End-to-end CFD pipeline achieved