

Extending the CFDof Add-on in FreeCAD

Anugya Chaubey

Semester-Long Internship 2026
OpenFOAM GUI

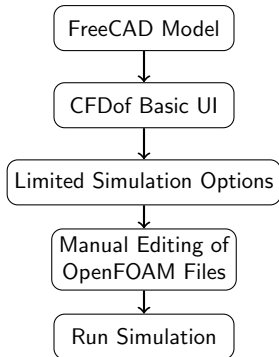
March 7, 2026

- 1 Problem Statement
- 2 Background / Context
- 3 Objectives
- 4 Proposed Solution / Approach
- 5 Timeline & Milestones
- 6 Deliverables
- 7 References

Problem Statement

- ◀ FreeCAD is an open-source CAD software used for 3D modeling and engineering design.
- ◀ It includes a CFD add-on called CFDof that allows users to run simulations using OpenFOAM.
- ◀ However, the current CFDof interface provides only limited access to OpenFOAM features.
- ◀ Many advanced simulation settings are not available through the UI.
- ◀ Users often need to manually edit OpenFOAM configuration files.
- ◀ This makes the workflow complicated and difficult for beginners.
- ◀ The goal of this project is to extend the CFDof add-on and expose more OpenFOAM functionality through the UI.

Current CFDof Workflow (Existing System)



Background

- ◀ Computational Fluid Dynamics (CFD) is used to simulate fluid flow, heat transfer and turbulence in engineering systems.
- ◀ OpenFOAM is a powerful open-source CFD toolkit used for these simulations.
- ◀ However, OpenFOAM mainly works through command-line tools and configuration files.
- ◀ This makes it difficult for beginners to configure simulations correctly.
- ◀ FreeCAD provides a graphical interface for 3D modeling.
- ◀ The CFDof add-on connects FreeCAD models with OpenFOAM solvers.

Background (Current Features)

- ◀ The current CFDof add-on already provides several useful features:
 - Geometry import from FreeCAD models
 - Mesh generation using OpenFOAM meshing tools
 - Basic solver setup (steady and transient solvers)
 - Boundary condition assignment through the UI
 - Running OpenFOAM simulations directly from FreeCAD
- ◀ These features help users create domains, generate meshes and run basic CFD simulations.
- ◀ However, advanced OpenFOAM capabilities such as dynamic mesh configuration are not available in the UI.
- ◀ For example, enabling dynamic mesh simulations requires manually editing the `dynamicMeshDict` file in OpenFOAM, which makes the workflow more complex.

Objectives

- ◀ Extend the CFDof interface to expose additional OpenFOAM simulation settings directly in FreeCAD.
- ◀ Implement UI support for selecting and configuring different turbulence models.
- ◀ Add UI controls for configuring numerical discretization schemes used by OpenFOAM solvers.
- ◀ Introduce configuration support for dynamic mesh simulations.
- ◀ Add UI options for basic multiphase simulation setup.
- ◀ Ensure that all selected options automatically generate the correct OpenFOAM configuration files.

Proposed Solution

- ◀ The project will extend the existing CFDof add-on by modifying the UI components and the OpenFOAM configuration generation system.
- ◀ Four major features will be implemented:
- ◀ **Numerical Scheme Configuration**
 - Users will be able to select discretization schemes for gradient, divergence and laplacian operations.
 - These selections will automatically update the `fvSchemes` configuration file.
- ◀ **Turbulence Model Selection**
 - A dropdown menu will allow users to select turbulence models such as k-epsilon or k-omega SST.
 - The chosen model will update the `turbulenceProperties` file used by OpenFOAM.

Proposed Solution (Implementation)

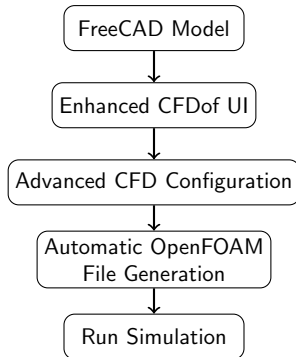
◀ Dynamic Mesh Support

- A UI panel will allow users to enable dynamic mesh simulations.
- The interface will generate and configure the `dynamicMeshDict` file automatically.
- This enables simulations involving moving boundaries or mesh deformation.

◀ Multiphase Flow Setup

- The interface will provide options for selecting multiphase solvers.
 - Users will be able to define fluid properties for each phase.
 - The UI will automatically generate configuration files such as `transportProperties`.
- ◀ These additions will allow advanced OpenFOAM simulations to be configured directly inside FreeCAD.

Improved CFDof Workflow (Proposed System)



Timeline & Milestones

◀ Weeks 1–2: Study and Preparation

- Study FreeCAD architecture and CFDof add-on structure
- Understand OpenFOAM configuration files and solver workflow
- Review documentation and research material related to CFDof

◀ Weeks 3–4: UI Design and Planning

- Identify missing features in the current CFDof interface
- Design UI layout for new configuration panels
- Plan how UI selections will map to OpenFOAM configuration files

◀ Weeks 5–6: Numerical Schemes Implementation

- Add UI controls for selecting discretization schemes
- Implement automatic generation of the fvSchemes file

◀ Weeks 7–8: Turbulence Model Integration

- Implement UI selection for turbulence models
- Configure turbulenceProperties based on user input

Timeline & Milestones (Continued)

◀ Weeks 9–10: Dynamic Mesh Support

- Create UI panel for enabling dynamic mesh simulations
- Generate and configure the `dynamicMeshDict` file

◀ Week 11: Multiphase Flow Setup

- Add UI options for multiphase solver configuration
- Implement generation of required OpenFOAM property files

◀ Week 12: Testing and Documentation

- Test the new features with sample simulation cases
- Fix issues and improve stability
- Prepare documentation and final report

Deliverables

- ◀ Extended CFDof add-on with additional UI features.
- ◀ Support for numerical schemes and advanced solver configuration.
- ◀ Implementation of turbulence models and multiphase simulations.
- ◀ Dynamic mesh capability integrated into the system.
- ◀ Support for phase change and mixing simulations.
- ◀ Documentation explaining system architecture and UI design decisions.
- ◀ Example simulation cases demonstrating the implemented features.

References

- ◀ OpenFOAM Foundation Documentation
<https://www.openfoam.com/documentation>
- ◀ OpenFOAM User Guide
<https://www.openfoam.com/documentation/user-guide>
- ◀ FreeCAD Official Documentation <https://wiki.freecad.org>
- ◀ CFDof / CfdOF FreeCAD Workbench Documentation
<https://github.com/jaheyms/CfdOF>
- ◀ Greenshields, C. J. (2022). *OpenFOAM User Guide*. CFD Direct.
<https://doc.cfd.direct/openfoam/user-guide>
- ◀ Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Pearson Education.