Streamlining Open-Source CFD: A Comparative Study of GUI-based vs. Script-based OpenFOAM Workflows

Medehal Pavan Sai Tarun[1], Mr. Vedant Dubey[2], Mr. Diptangshu Dey[3]

[1]VIT University, Bhopal

[2]National Institute of Technology, Raipur

[3]National Institute of Technology, Durgapur

OpenFOAM is a highly popular open-source CFD application widely used for computational fluid dynamics problems in aerospace and mechanical engineering. However, OpenFOAM's script-based operation using case dictionaries faces a major drawback for users. To improve user friendliness and to overcome this disadvantage of OpenFOAM's script-based operation using case dictionaries, several GUI applications such as the FreeCAD CfdOF add-on, Helyx-OS, SimFlow, and cfMesh-based GUI applications have been created. This research migration project focuses on migrating standard OpenFOAM CFD workflows from native script-based case setup to a GUI-based workflow using the FreeCAD CfdOF add-on, under the FOSSEE OpenFOAM GUI initiative. As part of this work, multiple OpenFOAM training cases were first executed using native OpenFOAM with manual definition of geometry, meshing, boundary conditions, and solver parameters. The same cases were then recreated using FreeCAD with the CfdOF add-on, which integrates CAD-based geometry creation, meshing, and solver setup while using OpenFOAM as the computational backend.In addition to training cases, an aerodynamic airfoil simulation was carried out using the FreeCAD CfdOF add-on to demonstrate the application of a GUI-based OpenFOAM workflow to an aerospace-relevant CFD problem. The work emphasizes ease of setup, workflow transparency, and reproducibility of OpenFOAM case files. The results show that the GUI-based workflow using FreeCAD CfdOF can reproduce standard OpenFOAM simulations while reducing user effort, supporting the adoption of OpenFOAM in CFD education.