OpenFOAM + WSL LaTeX Cheat Sheet

Jose Parra

August 8, 2025

1 WSL + Ubuntu Setup

```
# Install WSL and Ubuntu from Microsoft Store
wsl --install

# Check your WSL version
wsl --list --verbose

# Launch Ubuntu
Ubuntu 22.04
```

2 OpenFOAM Setup

```
# Add OpenFOAM repo and install (v11 example)
sudo apt update
sudo apt install curl
curl -s https://dl.openfoam.com/add-debian-repo.sh | sudo bash
sudo apt update
sudo apt install openfoam11-default

# Source OpenFOAM bashrc (one-time per session)
source /opt/openfoam11/etc/bashrc

# (Optional) Make permanent in ~/.bashrc
nano ~/.bashrc
# Add at the bottom:
source /opt/openfoam11/etc/bashrc
```

3 Run a Cavity Case

```
# Set up working directory
mkdir -p $FOAM_RUN
cp -r $FOAM_TUTORIALS/incompressibleFluid/cavity $FOAM_RUN
cd $FOAM_RUN/cavity

# Generate mesh
blockMesh

# If icoFoam is unavailable, switch to pisoFoam:
nano system/controlDict
# Change:
application pisoFoam;

# Then run the solver
pisoFoam
```

4 ParaView Visualization

```
# Option A: Use ParaView on Windows
foamToVTK

# Navigate in File Explorer:
\\wsl.localhost\Ubuntu-22.04\home\<username>\OpenFOAM\<user>-11\run\cavity\VTK
# Open .vtk files in Windows ParaView


# Option B: If using WSLg and paraFoam works
paraFoam
```

5 Common OpenFOAM Solvers

- **pisoFoam** - Transient, incompressible flow (use instead of icoFoam)

- **simpleFoam** - Steady-state incompressible solver

- **potentialFoam** - Potential flow solver (for initial guess)

6 Quick Tests

```
# Check if solver is installed
which pisoFoam

# Check environment variables
echo $FOAM_RUN

# List available solvers
foamListApps | grep Foam
```

7 Misc Tips

- Use 'nano' or 'emacs' to edit files like controlDict

- Save and quit nano: Ctrl+O, Enter, Ctrl+X

- Save and quit emacs: Ctrl+X, Ctrl+S; then Ctrl+X, Ctrl+C

- Use 'foamToVTK' to export to ParaView if 'paraFoam' doesn't work

- Files reset if using Live USB (install Ubuntu to make changes persist)