



Instruction Sheet for OpenFOAM v2412 for Ubuntu 22.04

Bioinspired Fluid-Structure Interaction Laboratory

Aerospace Engineering

University of Birmingham

Overview

This sheet explains how to install **OpenFOAM v2412** (OpenCFD/ESI distribution) on Ubuntu and verify the installation by running the `pitzDaily` tutorial. It mirrors the structure of the OpenFOAM 9 installation handout used previously, with updated repositories, packages, and environment setup for v2412.

Supported Ubuntu versions: OpenCFD lists precompiled packages for Ubuntu 24.10, 24.04 LTS, 22.04 LTS (and others).

Note (Windows/WSL): If you use **Windows Subsystem for Linux (WSL2)**, open the Ubuntu terminal in WSL and follow *exactly* the same steps below (admin privileges are required for adding repositories and installing packages).

Before you begin

- Ensure a stable internet connection; package downloads are large. (Same caution as the original sheet.)
- You will need `sudo` privileges to add repositories and install packages.

1. Open a terminal

Press `Ctrl+Alt+T` to open a terminal in Ubuntu. (On Windows/WSL2: open the Ubuntu app.)

2. Add the OpenCFD ([openfoam.com](https://dl.openfoam.com)) repository

- Copy and paste the following command into your **terminal prompt** to add the signing key and Debian/Ubuntu repository for OpenCFD/ESI builds, then update package lists:

```
curl -s https://dl.openfoam.com/add-debian-repo.sh | sudo bash
```

or

```
wget -q -O - https://dl.openfoam.com/add-debian-repo.sh | sudo bash
```

- Update the apt package list to account for the new download repository location by typing the following in terminal prompt.



```
sudo apt-get update
```

3. Install OpenFOAM v2412

Install OpenFOAM 2412 which also installs ParaView as a dependency by typing the following in terminal prompt.

```
sudo apt-get install openfoam2412-default
```

Optional: Install an editor & ParaView

```
sudo apt-get install gedit  
sudo apt-get install paraview
```

4. Source OpenFOAM in your shell

- Open the .bashrc file in the user home directory in an editor, e.g. by typing the following in terminal prompt. (**note the dot**)

```
gedit ~/.bashrc
```

Note: if gedit isn't working, please use notepad.exe /.bashrc

- Add the following line at the *end* of /.bashrc:

```
alias of2412='source /usr/lib/openfoam/openfoam2412/etc/bashrc '
```

Note: If you already have OpenFOAM-9 installed on your system, you can use the 'alias' feature to switch between OpenFOAM-9 and OpenFOAM-2412. Replace the following line at the end of the same bashrc file to make an alias for OpenFOAM-9.

```
alias of9='source /opt/openfoam9/etc/bashrc '
```

Note: When using the alias feature, you will have to type **of9** to initialize OpenFOAM-9 environment or **of2412** to initialize OpenFOAM-2412 everytime you open the terminal.

- Save and close the bashrc file, then reload it:

```
source ~/.bashrc
```



5. Verify the installation with **pitzDaily**

As in the original v9 sheet, we verify using **simpleFoam** on the **pitzDaily** tutorial. :contentReference[oaicite:11]index=11

1. Create a run area in your home and switch to it:

```
mkdir -p $FOAM_RUN  
cd $FOAM_RUN
```

2. Copy the tutorial case:

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .
```

3. Enter the case directory:

```
cd pitzDaily
```

4. Generate the mesh:

```
blockMesh
```

5. Run the steady incompressible solver:

```
simpleFoam
```

6. Visualize:

```
paraFoam
```

In ParaView, click **Apply** to load the dataset and view the geometry.

Note: If paraview does not open after typing the above command, install paraView using the command mentioned below

```
sudo apt-get install paraview
```

Troubleshooting notes

- If **openfoam2412** commands are not found, ensure your **/.bashrc** actually sources the **v2412** **bashrc**, then start a new terminal or run **source ~/.bashrc**.
- If repository signing or availability errors occur during **apt** operations, re-run the repository setup (Step 2) or try a bit later.