Install OpenFOAM 13 on Windows 10/11 using WSL

Beginner-friendly installation notes for OpenFOAM 13 on Windows 10/11 using WSL and Ubuntu 22.04

Prepared for Simwize Engineering Simulation Resources

Before starting: which terminal should you use?

PowerShell is the Windows terminal used here for WSL installation commands. Open it from Windows as Administrator.

CMD is also a Windows terminal, but it is different from **PowerShell**. In these notes, use **PowerShell** whenever a step says **PowerShell**.

Ubuntu terminal / Bash is the Linux terminal inside WSL. The Linux/OpenFOAM commands in this guide are run inside Ubuntu/WSL after Ubuntu has been installed.

A) Windows side - PowerShell as Administrator

1. Install WSL with Ubuntu 22.04

Use: PowerShell as Administrator

```
wsl --install -d Ubuntu-22.04
```

This installs WSL and Ubuntu 22.04. Restart Windows after the installation if requested.

After the installation, Ubuntu may ask you to create a Linux username and password. This username and password are for Ubuntu/WSL and do not need to be the same as your Windows login.

2. Open Ubuntu / WSL terminal

Use: PowerShell

```
wsl -d Ubuntu-22.04
```

Alternatively, open Ubuntu 22.04 from the Windows Start menu.

Ubuntu terminal / Bash is the Linux terminal inside WSL. The next commands should be run inside this Ubuntu/WSL terminal.

B) Linux side - Ubuntu terminal / Bash

3. Check Ubuntu version

Use: Ubuntu terminal / Bash

```
lsb_release -a
```

This confirms that Ubuntu 22.04 is installed.

4. Update Ubuntu

Use: Ubuntu terminal / Bash

```
sudo apt update && sudo apt upgrade -y
```

This refreshes the package information and installs available updates.

5. Install required helper tools

Use: Ubuntu terminal / Bash

```
sudo apt install -y wget software-properties-common
```

This installs the tools needed to add the OpenFOAM repository.

6. Add the OpenFOAM public key

Use: Ubuntu terminal / Bash

```
sudo sh -c "wget -O - https://dl.openfoam.org/gpg.key >
/etc/apt/trusted.gpg.d/openfoam.asc"
```

This allows Ubuntu to verify OpenFOAM packages.

7. Add the OpenFOAM repository

Use: Ubuntu terminal / Bash

```
sudo add-apt-repository http://dl.openfoam.org/ubuntu
```

This tells Ubuntu where to download OpenFOAM from.

8. Update the package list again

Use: Ubuntu terminal / Bash

```
sudo apt update
```

This loads the package list from the OpenFOAM repository.

9. Install OpenFOAM 13 and ParaView

Use: Ubuntu terminal / Bash

```
sudo apt install -y openfoam13
```

This installs OpenFOAM 13. ParaView is usually installed as a dependency.

10. Add OpenFOAM to .bashrc

Use: Ubuntu terminal / Bash

```
notepad.exe ~/.bashrc
```

This opens the .bashrc file with Windows Notepad.

Add this line at the bottom of the .bashrc file:

```
. /opt/openfoam13/etc/bashrc
```

This activates OpenFOAM automatically whenever the Ubuntu terminal starts. There must be a space after the dot.

You can also open .bashrc directly using any text editor. The general location of this file is:

```
/home/<your-username>/.bashrc
```

If you do not see the file in this folder, enable Show hidden files, because files starting with a dot are hidden by default. After saving the .bashrc file, close and reopen the Ubuntu terminal.

11. Optional: create aliases for different OpenFOAM versions

Aliases are useful when more than one OpenFOAM version is installed on the same system. Add alias lines at the bottom of the same .bashrc file, for example:

```
alias of13='source /opt/openfoam13/etc/bashrc'

alias of12='source /opt/openfoam12/etc/bashrc'
```

This lets you quickly switch to a specific OpenFOAM version by typing the corresponding alias in the Ubuntu terminal. For example:

```
of13
```

After saving the .bashrc file, close and reopen the Ubuntu terminal.

12. Optional: select the correct GPU for ParaView

If ParaView crashes or behaves unexpectedly, the issue may be related to graphics adapter selection, especially on systems with more than one GPU.

In that case, WSL graphical applications can be instructed to use a specific graphics adapter by adding the following line at the bottom of the same .bashrc file:

```
export MESA_D3D12_DEFAULT_ADAPTER_NAME=<adapter-name>
```

Replace <adapter-name> with a suitable part of your graphics adapter name, for example NVIDIA, AMD, Radeon, or Intel, depending on your system.

Example:

```
export MESA_D3D12_DEFAULT_ADAPTER_NAME=NVIDIA
```

This can help ParaView use the correct graphics adapter instead of the default one selected automatically by WSL. After saving the .bashrc file, close and reopen the Ubuntu terminal before starting ParaView again.

13. Check OpenFOAM

Use: Ubuntu terminal / Bash

```
foamVersion
```

This confirms that OpenFOAM is installed and the OpenFOAM environment is loaded correctly.

Expected output:

```
OpenFOAM-13
```

14. Optional: use software rendering for ParaView

If ParaView still does not open correctly, you can try software rendering:

```
LIBGL_ALWAYS_SOFTWARE=1 paraview
```

This starts ParaView using CPU/software rendering instead of the GPU rendering path.

If this command works on your system, the same setting can be added to .bashrc so that it is applied automatically in every new Ubuntu/WSL session.