



Instruction Sheet for OpenFOAM v2412 for Windows OS

Bioinspired Fluid-Structure Interaction Laboratory

Aerospace Engineering

University of Birmingham

Overview

This sheet explains how to install **OpenFOAM v2412** (OpenCFD/ESI distribution) on windows OS.

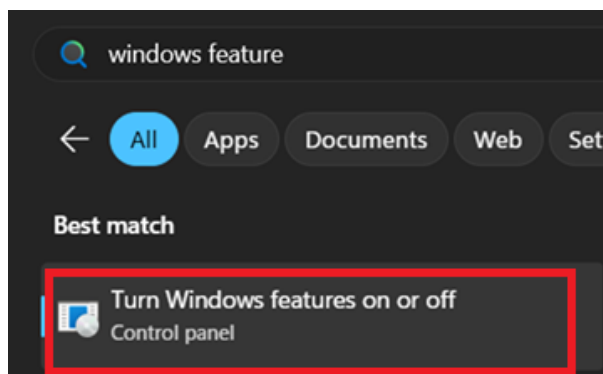
Before you begin

- Ensure a stable internet connection; package downloads are large. (Same caution as the original sheet.)

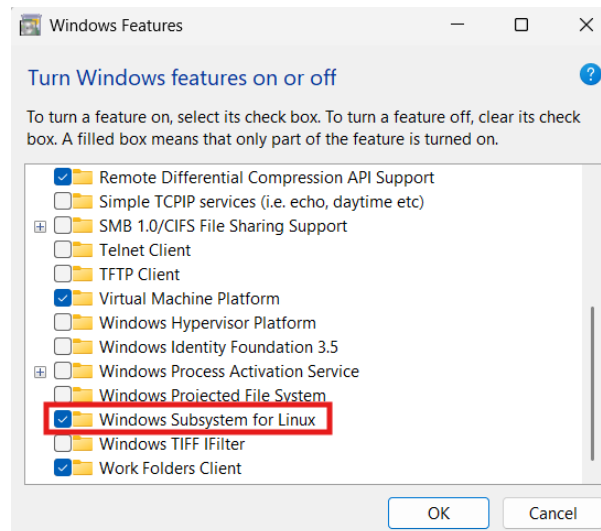
1. Install Windows Subsystem for Linux (WSL) with Ubuntu

1. Install WSL2

- (a) In windows, search "Turn windows feature on or off"



- (b) Open it and scroll down to the bottom. Find "Windows Subsystem for Linux" option and click it.

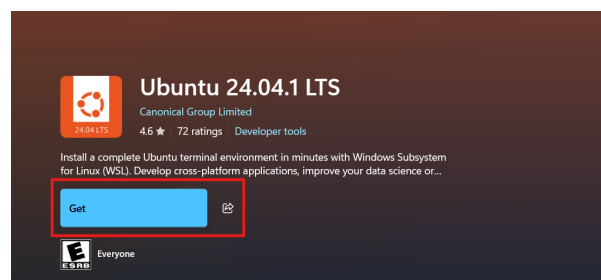


(c) Restart your PC to apply changes.

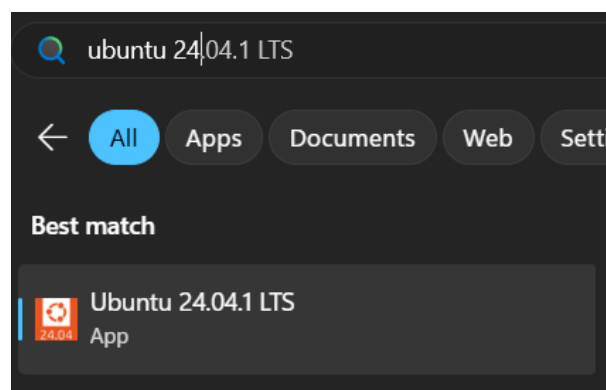
2. Install Ubuntu from Microsoft Store

(a) Go to "Microsoft Store" and search "Ubuntu"

(b) Install the latest version of Ubuntu.



(c) Search Ubuntu in search bar and open it.



(d) Set your LINUX username and password.



```
karki@Yukesh: ~  
Installing, this may take a few minutes...  
Please create a default UNIX user account. The username does not need to match your Windows username.  
For more information visit: https://aka.ms/wslusers  
Enter new UNIX username: karki  
New password:  
Retype new password:  
passwd: password updated successfully  
Installation successful!  
To run a command as administrator (user "root"), use "sudo <command>".  
See "man sudo_root" for details.  
  
Welcome to Ubuntu 24.04.1 LTS (GNU/Linux 5.15.133.1-microsoft-standard-WSL2 x86_64)  
  
 * Documentation:  https://help.ubuntu.com  
 * Management:    https://landscape.canonical.com  
 * Support:       https://ubuntu.com/pro  
  
System information as of Fri Sep 26 10:38:10 BST 2025  
  
System load:  0.84      Processes:            59  
Usage of /:   0.1% of 1006.85GB   Users logged in:     0  
Memory usage: 5%      IPv4 address for eth0: 192.168.204.43  
Swap usage:   0%  
  
This message is shown once a day. To disable it please create the  
/home/karki/.hushlogin file.  
karki@Yukesh:~$ |
```

2. Install OpenFOAM v2412

- (a) Go to website: www.openfoam.com
- (b) Go to Download -> Linux -> Debian+ Ubuntu
- (c) Open WSL terminal from search bar
- (d) Copy the given command one by one:
 - (i) Add repository

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

```
karki@Yukesh:~$ curl -s https://dl.openfoam.com/add-debian-repo.sh | sudo bash|
```

- (ii) Update repository information

```
sudo apt-get update
```

```
karki@Yukesh:~$ sudo apt-get update|
```

- (iii) Install OpenFOAM

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

```
karki@Yukesh:~$ sudo apt-get install openfoam2412-default|
```



Press Y and enter.

```
karki@Yukesh:~$ sudo apt-get install openfoam2412-default
Reading package lists... Done
Building dependency tree
Reading state information... Done
The following additional packages will be installed:
  openfoam2412 openfoam2412-common openfoam2412-dev openfoam2412-source openfoam2412-tools openfoam2412-tutorials
The following NEW packages will be installed:
  openfoam2412 openfoam2412-common openfoam2412-default openfoam2412-dev openfoam2412-source openfoam2412-tools
  openfoam2412-tutorials
0 upgraded, 7 newly installed, 0 to remove and 62 not upgraded.
Need to get 114 MB of archives:
After this operation, 640 MB of additional disk space will be used.
Do you want to continue? [Y/n]
```

(e) Source OpenFOAM to start using it.

(i) Copy the path.

```
karki@Yukesh: ~
Selecting previously unselected package openfoam2412-tutorials.
Preparing to unpack .../5-openfoam2412-tutorials_2412.250528-1_all.deb ...
Unpacking openfoam2412-tutorials (2412.250528-1) ...
Selecting previously unselected package openfoam2412-default:amd64.
Preparing to unpack .../6-openfoam2412-default_2412.250528-1_amd64.deb ...
Unpacking openfoam2412-default:amd64 (2412.250528-1) ...
Setting up openfoam2412-source (2412.250528-1) ...
Setting up openfoam2412-common (2412.250528-1) ...
Setting up openfoam2412-tools:amd64 (2412.250528-1) ...
Setting up openfoam2412-tutorials (2412.250528-1) ...
Setting up openfoam2412:amd64 (2412.250528-1) ...
Update installation directory for openfoam2412
Configuring OpenFOAM (/usr/lib/openfoam/openfoam2412)
  Replaced WM_PROJECT_DIR by "/usr/lib/openfoam/openfoam2412" (etc/bashrc)
  Replaced WM_PROJECT_DIR by "/usr/lib/openfoam/openfoam2412" (etc/cshrc)
  Remove automatic projectDir setting (etc/bashrc)
  Remove automatic projectDir setting (etc/cshrc)
Registered 'openfoam2412' with openfoam-selector
**
** Different ways to setup OpenFOAM (www.openfoam.com)
** - shell session: /usr/bin/openfoam2412
** - select for login shell: openfoam-selector --set openfoam2412
** - source from ~/.bashrc file: . /usr/lib/openfoam/openfoam2412/etc/bashrc
**
** Upgrade information: www.openfoam.com/documentation/
**
Setting up openfoam2412-dev:amd64 (2412.250528-1) ...
Setting up openfoam2412-default:amd64 (2412.250528-1) ...
Processing triggers for man-db (2.9.1-1) ...
karki@Yukesh:~$
```

(ii) Open bashrc file.

```
nano ~/.bashrc
```

```
karki@Yukesh:~$ nano ~/.bashrc
```

(iii) Scroll down to the bottom and paste the copied path.

```
. /usr/lib/openfoam/openfoam2412/etc/bashrc
```



```
GNU nano 4.8 /home/karki/.bashrc

if [ -f ~/.bash_aliases ]; then
    . ~/.bash_aliases
fi

# enable programmable completion features (you don't need to enable
# this, if it's already enabled in /etc/bash.bashrc and /etc/profile
# sources /etc/bash.bashrc).
if ! shopt -oq posix; then
    if [ -f /usr/share/bash-completion/bash_completion ]; then
        . /usr/share/bash-completion/bash_completion
    elif [ -f /etc/bash_completion ]; then
        . /etc/bash_completion
    fi
fi

. /usr/lib/openfoam/openfoam2412/etc/bashrc
```

(iv) Save the file and exit.

Ctrl+o
Ctrl+x

(f) Close the WSL window and reopen it to use OpenFOAMv2412.