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.
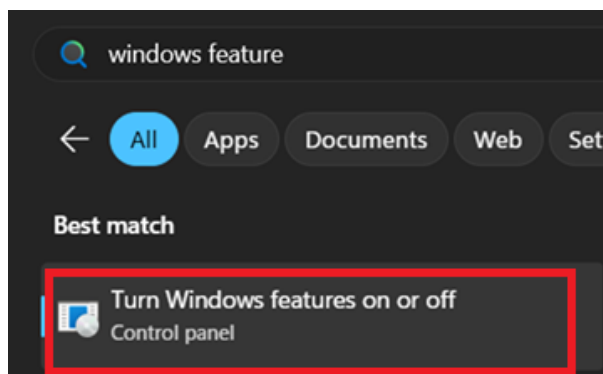
Before you begin

- Ensure a stable internet connection; package downloads are large. (Same caution as the original sheet.)
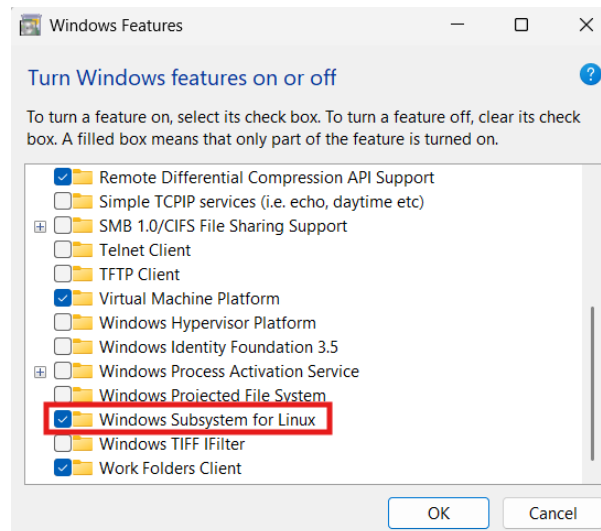
1. Install Windows Subsystem for Linux (WSL) with Ubuntu

1. Install WSL2

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

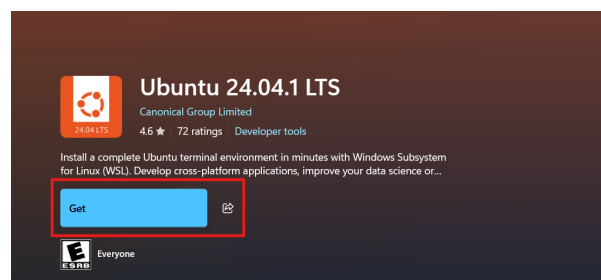(b) Open it and scroll down to the bottom. Find "Windows Subsystem for Linux" option and click it.

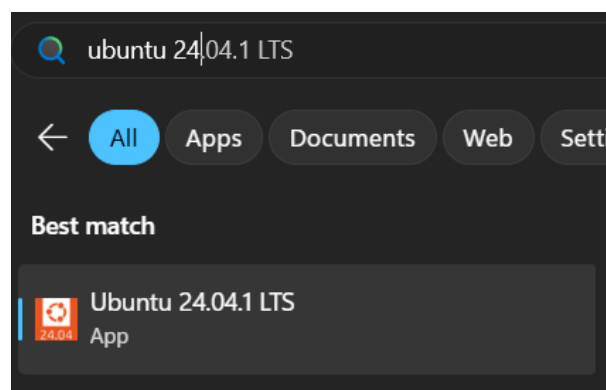(c) Restart your PC to apply changes.

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.

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
```


   (ii) Update repository information

```
sudo apt-get update
```


   (iii) Install OpenFOAM

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

Press Y and enter.

(e) Source OpenFOAM to start using it.

   (i) Copy the path.

   (ii) Open bashrc file.

```
nano ~/.bashrc
```


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

```
. /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.