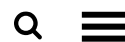


**OpenFOAM**

The OpenFOAM Foundation



OpenFOAM for Windows

The packaged distributions of **OpenFOAM for Ubuntu** can now be installed directly on Microsoft Windows 10 using **Windows Subsystem for Linux** (WSL). WSL provides a full compatibility layer for running Linux applications on Windows by performing real-time translation of Linux system calls into Windows OS system calls. The system can support graphical Linux applications, such as the version of *ParaView* that includes the OpenFOAM reader module, with additional X server software (see below). Running OpenFOAM applications in parallel using WSL is reported to work effectively.

Note: We do not support older versions of Windows, e.g. 7 or 8, because **Microsoft does not support them.**

Activate Windows Subsystem for Linux

- Follow the Guide to **Install WSL** and install the **Ubuntu 20.04 LTS** Linux Distribution.
- Make a note of the WSL version you are running, either **v1** or **v2**.
- Launch the Ubuntu distribution through WSL.

Installing OpenFOAM

Manage Cookie Consent



This website uses technologies like cookies to function properly. There are some technologies on the site which process data such as browsing behaviour or unique IDs. If you do not wish to give consent to process this data, click "Deny".

[Accept](#)[Deny](#)[Cookie Policy](#) [Privacy Policy](#)

```
sudo sh -c "wget -O - http://dl.openfoam.org/gpg.key | apt-key add -"  
sudo add-apt-repository "http://dl.openfoam.org/ubuntu dev"  
sudo apt-get update  
sudo apt-get install openfoam-dev
```

Compilation Tools

In order to compile applications and libraries in OpenFOAM, the user should install additional compilation tools by the following command:

```
sudo apt-get install build-essential
```

User Configuration

In order to use the installed OpenFOAM package, the user needs to set their environment for OpenFOAM as follows.

1. **One Time Only:** At the bottom of the user's `.bashrc` file, source the `bashrc` file in the OpenFOAM installation which contains the environment settings. For `openfoam10`, the following command avoids the need to open an editor (for OpenFOAM-dev, replace `openfoam10` with `openfoam-dev`):

```
echo ". /opt/openfoam10/etc/bashrc" >> $HOME/.bashrc
```

2. **One Time Only:** register the change to the `.bashrc` file by typing at the terminal prompt (note the dots):

```
. $HOME/.bashrc
```

Manage Cookie Consent



This website uses technologies like cookies to function properly. There are some technologies on the site which process data such as browsing behaviour or unique IDs. If you do not wish to give consent to process this data, click "Deny".

Accept

Deny

[Cookie Policy](#) [Privacy Policy](#)

Enabling Graphical Applications

To run graphical Linux applications, such as *ParaView* or the *gedit* editor, requires the installation of X server software. The most popular X server software for Windows is **VcXsrv**, which can be installed as described below.

- Download the **VcXsrv installer**.
- (if the above link does not work, go to the **VcXsrv files page** and download the latest `vcxsrv-64.X.X.X.X.installer.exe` file).
- Run *XLaunch* that was installed by VcXsrv.
- Open the “Extra settings” window and: a) Deselect (uncheck) “Native opengl”; b) Select “Disable access control”.

When a `bash` shell is opened, the `DISPLAY` environment variable needs to point to the X server that is running. To make this addition permanent, set the `DISPLAY` in the user’s `.bashrc` file by the following command:

- for WSL v1

```
echo "export DISPLAY=:0" >> ${HOME}/.bashrc
```

- for WSL v2

```
echo "export DISPLAY=$(awk '/nameserver / {print $2; exit}' /etc/resolv.conf
```

Source the `.bashrc` file again, i.e. execute **one time only**:

Manage Cookie Consent



This website uses technologies like cookies to function properly. There are some technologies on the site which process data such as browsing behaviour or unique IDs. If you do not wish to give consent to process this data, click “Deny”.

Accept

Deny

[Cookie Policy](#) [Privacy Policy](#)

Try to start *ParaView* from a terminal by typing:

```
paraview
```

There is a bug in **WSL v1 only** which may cause ParaView to fail to open with the message:

```
/opt/paraviewopenfoam56/lib/paraview: error while loading shared libraries: libQt5Core.so.5: cannot open shared object file: No such file or directory
```

If this occurs, enter the following commands (copy/paste) in the terminal:

```
sudo cp /usr/lib/x86_64-linux-gnu/libQt5Core.so.5 /opt/paraviewopenfoam56/mesa/lib/
sudo strip --remove-section=.note.ABI-tag /opt/paraviewopenfoam56/mesa/lib/libQt5Core.so.5
```

Next Steps

See [OpenFOAM on Ubuntu: Getting Started](#). If the user has enabled graphical applications, they can open the *gedit* editor in the background (&) with

```
gedit &
```

Otherwise without graphical support, 3 popular editors which can work through a terminal are:

- **nano**: the easiest of the 3 editors for the purpose, see [nano basics guide](#);
- **emacs**: powerful editor that uses a more complex set of key commands, [emacs basics](#);
- **vim**: another editor with arguably a less familiar set of key commands, see [vim quick guide](#).

Manage Cookie Consent



This website uses technologies like cookies to function properly. There are some technologies on the site which process data such as browsing behaviour or unique IDs. If you do not wish to give consent to process this data, click "Deny".

Accept

Deny

[Cookie Policy](#) [Privacy Policy](#)



LATEST NEWS

OpenFOAM & ParaView 5.10

19th January 2023

Funding OpenFOAM in 2023

8th November 2022

OpenFOAM v10 | Patch Releases

28th July 2022

OpenFOAM 10 Released

12th July 2022

Download v10 | Ubuntu

12th July 2022

Download v10 | Linux

12th July 2022

ARTICLES

Modular Solvers in OpenFOAM

Manage Cookie Consent



This website uses technologies like cookies to function properly. There are some technologies on the site which process data such as browsing behaviour or unique IDs. If you do not wish to give consent to process this data, click "Deny".

Accept

Deny

[Cookie Policy](#) [Privacy Policy](#)

RECENT TWEETS

RT @CFDdirect: New filmSurfaceVelocity boundary condition evaluates a film surface velocity from the shear imposed by the neighbouring fluid...2 days ago

RT @CFDdirect: Subscribe to our newsletter for the latest #OpenFOAM developments, releases, documentation, videos, cloud, training: <https://openfoam.org/newsletter/>...2 days ago

RT @CFDdirect: The mappedFilmPressure boundary condition maps the pressure from a fluid region to a film region in #OpenFOAM-dev <https://openfoam.org/docs/latest/mappedFilmPressure/>...3 days ago

The OpenFOAM Foundation Ltd
Incorporated in England
Company No. 9012603
VAT Reg No. GB 211 0914 63

Directors:
Henry Weller
Chris Greenshields
Cristel de Rouvray

Address:
PO Box 56676
London, W13 3DB
United Kingdom

Legal:

Manage Cookie Consent



This website uses technologies like cookies to function properly. There are some technologies on the site which process data such as browsing behaviour or unique IDs. If you do not wish to give consent to process this data, click "Deny".

Accept

Deny

[Cookie Policy](#) [Privacy Policy](#)