# Guide for Installing OpenFoam in Windows

*S.M.Sudharsan*

Installing OpenFoam in windows could be done in three major steps, which are:

1. Enabling “Hardware Assisted Virtualization”
2. Installing “blueCFD-core” for windows
3. Installing “OpenFoam” for windows

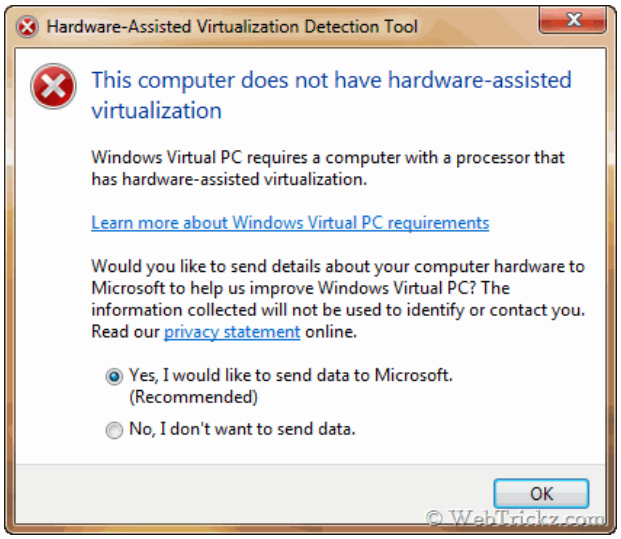
While there are separate tutorials and videos available on installation of these software, there is no single documentation compiling all the steps and this document is aimed at achieving this.

## 1. Enabling “Hardware Assisted Virtualization”

Virtualization solutions enable you to run multiple Virtual Machines (VM) on a single PC. In most of the windows systems, it will be disabled by default. For proceeding further with the installation this needs to be turned on. To check the status of it, install “Microsoft Hardware-Assisted Virtualization Detection Tool” .

### *Installing and running “Microsoft Hardware-Assisted Virtualization Detection Tool”:*

- Download the application (havdetectiontool.exe) to your local machine. There is only one installer for both 32 bit and 64 bit machines.
- Double click on the application and follow the prompts. Note: The tool requires administrator privileges.
- Based on the condition of the virtualization setting one of the below output will be shown:


If Virtualization is turned off in your system, go into Bios settings and turn it on before further proceeding with the installation.

## *Enabling virtualization extensions in BIOS:*

- Reboot the computer and open the system's BIOS menu. This can usually be done by pressing the delete key, the F1 key or Alt and F4 keys depending on the system.
- Select Restore Defaults or Restore Optimized Defaults, and then select Save & Exit.
- Power off the machine and disconnect the power supply.
- Power on the machine and open the BIOS (as per Step 1).
- Open the Processor submenu, the processor settings menu may be hidden in the Chipset, Advanced CPU Configuration or Northbridge.
- Enable Intel Virtualization Technology (also known as Intel VT) or AMD-V depending on the brand of the processor. The virtualization extensions may be labeled Virtualization Extensions, Vanderpool or various other names depending on the OEM and system BIOS.
- Enable Intel VTd or AMD IOMMU, if the options are available. Intel VTd and AMD IOMMU are used for PCI pass-through.
- Select Save & Exit.
- Power off the machine and disconnect the power supply.
- Run cat /proc/cpuinfo | grep vmx svm. If the command outputs, the virtualization extensions are now enabled. If there is no output your system may not have the virtualization extensions or the correct BIOS setting enabled.

For further details on Bios part look here.

## 2. Installing "blueCFD-core" for windows

blueCFD is an open source project with the primary objective being to bring OpenFoam technology to run natively in Windows operating systems. blueCFD allows us to compile from source code and install OpenFoam technology on Microsoft Windows as done in Linux distributions. This is achieved by relying on MSys2 as a development infrastructure.

### *Installation procedure:*

The software could be downloaded from here.

The above link provides the .exe file of the software. Once it is downloaded, run it and agree to the terms and conditions of the software. Following which you will be asked for installation location, go with the default folder or change if you wish. After this, do not change any default settings and continue with the installation. This package installs:

- blueCFD-core 2017-1
- ParaView 5.4.1
- MS-MPI 7.1
  These were the latest versions when this doc was created. Click here for downloading future versions.

A brief video is also available showing the installation procedure of the software, look at it for any further clarifications.

With this, the installation is done and now your system is ready to run OpenFoam simulations, and view/ edit them as in Linux systems.
The blueCFD acts like a virtual Linux terminal and the commands used in Linux could be directly used here. ParaView helps with the viewing and post-processing stages.

## 3. Installing "OpenFoam" for windows

Last but not the least, we need to install OpenFoam executable for windows which is available here.

Once it is downloaded follow the instructions given below to install:

- Run the OpenFoam-installer executable. It will install Docker (including VirtualBox, Git), the precompiled v1712 OpenFoam executable image and creates the working environment. (Please note that this process can take several minutes depending on your system memory.)

- When the installation is complete, please click on the "OF_Create_Env" shortcut on the Desktop. A shell will open and close automatically, completing setup of the user working environment. This step is mandatory only on the first occasion of OpenFoam for Windows installation.

Now the OpenFoam along with other software are successfully installed and is ready to use. To run/create a simulation, open blueCFD-core terminal and start typing in the code. The visualization could be done in ParaView. The software now works as if it is installed in a Linux environment, so the same set of codes could be used and the tutorials available for Linux could be used for learning. Examples and tutorials could be found inside the blueCFD folder.

For further help on installation of OpenFoam, click here.

Useful Links and References:


1. https://docs-old.fedoraproject.org/en-US/Fedora/13/html/Virtualization_Guide/sect-Virtualization-Troubleshooting-Enabling_Intel_VT_and_AMD_V_virtualization_hardware_extensions_in_BIOS.html
2. http://bluecfd.github.io/Core/About/
3. https://www.microsoft.com/en-in/download/details.aspx?id=592
4. https://www.openfoam.com/download/txt/openfoam-windows-readme_v1712.txt
5. https://www.openfoam.com/download/install-binary-windows.php