

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

051176 - Computational Techniques for Thermochemical Propulsion  
Master of Science in Aeronautical Engineering

## Install OpenFOAM on Your Own Machine



Prof. Federico Piscaglia

Dept. of Aerospace Science and Technology (DAER)  
POLITECNICO DI MILANO - Italy

[federico.piscaglia@polimi.it](mailto:federico.piscaglia@polimi.it)



# Installing OpenFOAM on your laptop

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test



OpenFOAM is written for the UNIX and GNU/Linux operating systems, therefore installation on a native \*nix OS remains the best choice.

However, several options are available if a \*nix machine is not available:

- 1) a native Linux/Unix distribution on your Laptop  
→ you can install and start Linux from a bootable USB key!
- 2) a “containerized” version of OpenFOAM ([Docker](#), any OS<sup>1</sup>)  
→ Win10 only: Ubuntu bash shell
- 3) a Linux distribution running on a Virtual Machine ([VirtualBox](#), any OS<sup>1</sup>)

Moreover, you can decide to compile OpenFOAM from source or to install a precompiled version.

→ **we will consider first how to install and compile OpenFOAM from the source on a Linux (Ubuntu/Debian) machine**



# Post-processing: ParaView

Install

liveDistro

docker

system req.

install

Linux/Mac

win+bash

VirtualBox

OpenFOAM

Compile

Test



ParaView is an open-source software for scientific visualization. It is not part of OpenFOAM, but it is well integrated with it and represents the best choice for OpenFOAM graphical postprocessing.

Again, you can compile ParaView from source or download a binary build (**any OS**).

## Compile your own version ParaView if:

- You want to use the OpenFOAM reader supplied with OpenFOAM itself
- You want to use some non-default plugin
- You need a non-standard installation (e.g. without GUI)

## Use a binary package if:

- You are content with the builtin OpenFOAM reader
- You are OK with the standard set of readers and plugins
- You cannot/do not want to compile ParaView.



Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test



# Install a Linux live distribution



# Installing OpenFOAM on your laptop

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test



<https://tutorials.ubuntu.com/tutorial/tutorial-create-a-usb-stick-on-windows#0>

To install OpenFOAM on your Ubuntu Linux live distribution, see instruction about how to install OpenFOAM on Ubuntu.



Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test





# Docker for the class → fpisk/cttp

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

Screenshot of a Docker repository page for the fpisk/cttp image on a platform like DockerHub.

**PUBLIC REPOSITORY**

**fpisk/cttp**

Last pushed: 2 days ago

Repo Info Tags Collaborators Webhooks Settings

**Short Description**

Computational Techniques for Thermochemical Propulsion. Federico Piscaglia, Politecnico di Milano

**Full Description**

Docker Image of Ubuntu-18.04, Including all the necessary packages to compile and run OpenFOAM-6. The Image also includes all the supplementary material provided to the class "Computational Techniques for Thermochemical Propulsion" (Master of Science in Aerospace Engineering, Politecnico di Milano), a class on the CFD simulation of turbulent (multiphase) reactive flows in moving boundary problems with the open-source software OpenFOAM.

Prof. Federico Piscaglia - Dept. of Aerospace Science and Technology, Politecnico di Milano

**Docker Pull Command**

```
docker pull fpisk/cttp
```

**Owner**

fpisk

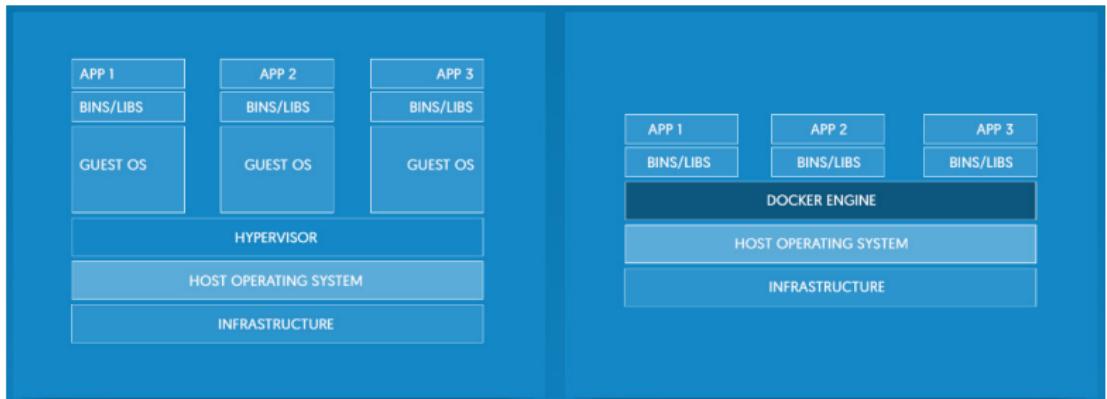
You can download an image of Ubuntu-18.04 LTS with OpenFOAM-7 installed, including the introductory material for the class.



# Using containers - Docker

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

Docker provides the ability to package and run an application in a loosely isolated environment called a container. The isolation and security allow you to run many containers simultaneously on a given host. **Containers are lightweight because they don't need the extra load of a hypervisor, but run directly within the host machine's kernel.** This means you can run more containers on a given hardware combination than if you were using virtual machines.



More info at:

<https://www.hptcbeginner.com/what-is-docker-docker-vs-virtualbox/>



# VirtualBox vs Docker/win10 bash

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test



## Docker is not a VM

- low disk requirement
- performance is reduced if compared to native Linux
- graphical applications are very slow (consider using native ParaView)
- almost no impact on performance on the host OS
- already available in Win10, otherwise must be installed

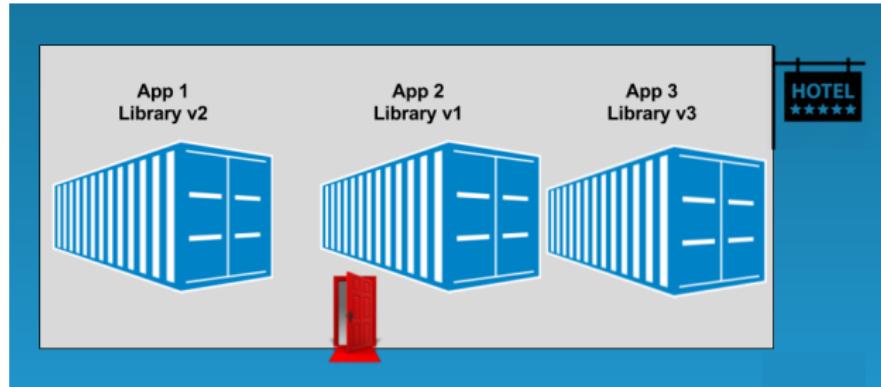
## The VM imposes a significant overhead on the host OS:

- performance is severely reduced
- host OS performance might be affected as well
- requires considerable additional disk space
- available for any OS



# Using containers - what is Docker?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test



- If a VM is a house then a docker container is a hotel room. If you don't like the setup, changing a hotel room is much easier than changing a house. A hotel has multiple rooms sharing the same underlying infrastructure (foundation, plumbing, electrical wires, etc.).
- **Docker offers the ability to run multiple applications within the same host OS**, sharing underlying resources (CPU, RAM, etc.).



# Install a Packaged Version of OpenFOAM

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

## PROS:

- Quick solution to start using OpenFOAM
- Very portable across Win/Linux/MacOS distributions
- Almost all the most significant versions of OF available

## CONS:

- Creating and packaging your own container is not trivial but...

... there are already packaged pre-compiled version of OpenFOAM in Docker available for you! **WHERE?**

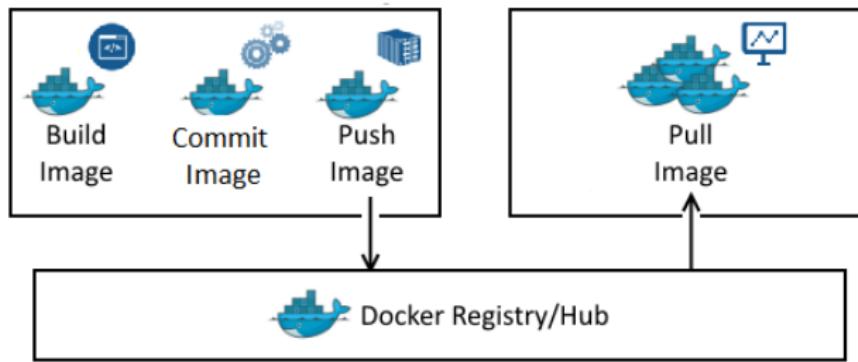
<https://hub.docker.com/r/fpisk/cttp/>

To download it, you need to log-in to Docker Hub and Docker Cloud [using your free Docker ID](#).



# DockerHub - hub.docker.com

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test



<https://hub.docker.com/>

**Docker Hub** is a cloud-based registry service which allows you to link to code repositories, build your images and test them, stores manually pushed images, and links to Docker Cloud so you can deploy images to your hosts.

- It provides a centralized resource for container image discovery, distribution and change management, user and team collaboration, and workflow automation throughout the development pipeline.
- Log in to Docker Hub and Docker Cloud using your free Docker ID.



# DockerHub - hub.docker.com

Install

liveDistro

docker

system req.

install

Linux/Mac

win+bash

VirtualBox

OpenFOAM

Compile

Test

New to Docker?  
Create your free Docker ID to get started.

Choose a Docker ID

Email address

Choose a password

I agree to Docker's [Terms of Service](#).  
 I agree to Docker's [Privacy Policy and Data Processing Terms](#).  
 I would like to receive email updates from Docker, including its various services and products.

Sign Up

© 2016 Docker Inc.

Make your own FREE account on Docker Hub and Docker Cloud at:

<https://hub.docker.com/>

you will get a **Docker ID** to login to the service.



# Docker for the class → fpisk/cttp

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

The screenshot shows a Docker repository page for the user 'fpisk'. The repository name is 'fpisk/cttp' with a star icon indicating it's public. It was last pushed 2 days ago. The page includes tabs for Repo Info, Tags, Collaborators, Webhooks, and Settings. The Repo Info section contains a 'Short Description' box with the text: 'Computational Techniques for Thermochemical Propulsion. Federico Piscaglia, Politecnico di Milano'. A 'Docker Pull Command' box contains the command 'docker pull fpisk/cttp'. The Full Description box provides a detailed explanation of the Docker image, stating it's based on Ubuntu-18.04 and includes OpenFOAM-6, supplementary material for a class, and information about Prof. Federico Piscaglia from Politecnico di Milano. The Owner section shows the profile picture of 'fpisk'.

You can download an image of Ubuntu-18.04 LTS with OpenFOAM-6 installed, including the introductory material for the class.



# Docker: system requirements

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

**FIRST, verify your Windows OS is 64-bit (x64)!**

If you have a 64-bit system, then you must ENABLE on your PC the following features:

- VT-x and Hyper-v
  - VT-x must be enabled in the BIOS, so you need to reboot your system and administrative rights.
  - Hyper-v is not available on Windows Home. Requires ‘VirtualBox’ to be installed in the Host OS. This is done automatically by Docker Desktop (formerly Docker Toolbox).
- windows options, to support:
  - Developer mode
  - Windows Hypervisor Platform (native or virtualized)
  - Subsystem for Linux (optional for docker)

You can find some detailed instructions to check your installation in the next slides or at this (official) link:

[https://docs.docker.com/toolbox/toolbox\\_install\\_windows/#step-3-verify-your-installation](https://docs.docker.com/toolbox/toolbox_install_windows/#step-3-verify-your-installation)

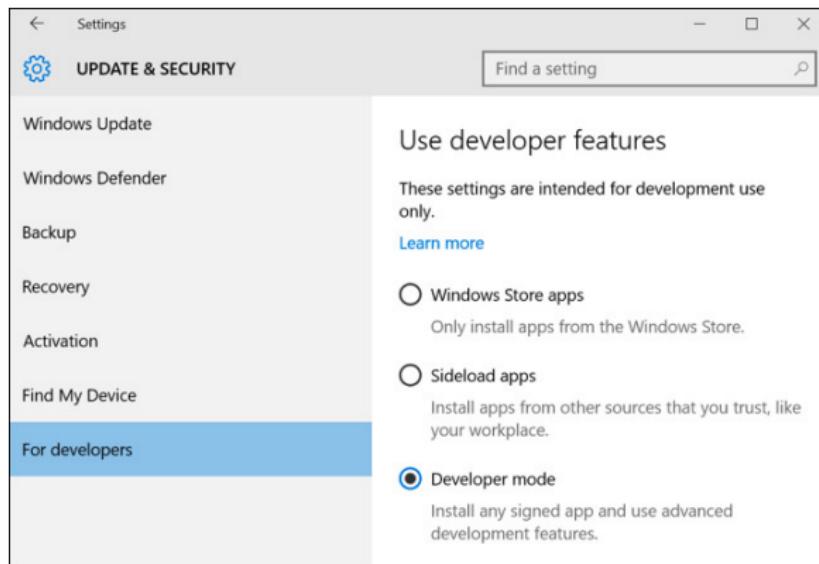


# Enable “Developer mode”

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

Starting from Windows 10 (64-bit only) it is possible to have a bash shell to run command-line Linux applications:

Settings → Update & Security → For Developers → activate “developer mode”

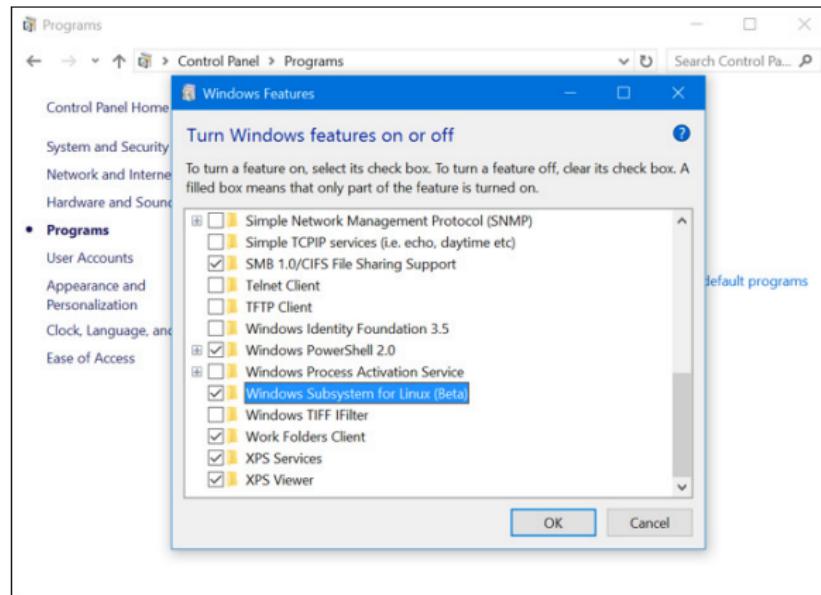




# Enable “Subsystem for Linux” option

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

Control panel → Programs → Turn Windows Features On or Off → Enable the “Windows Subsystem for Linux (Beta)” → OK. Reboot





# Enable the Windows Hypervisor Platform

Install

liveDistro

docker

system req.

install

Linux/Mac

win+bash

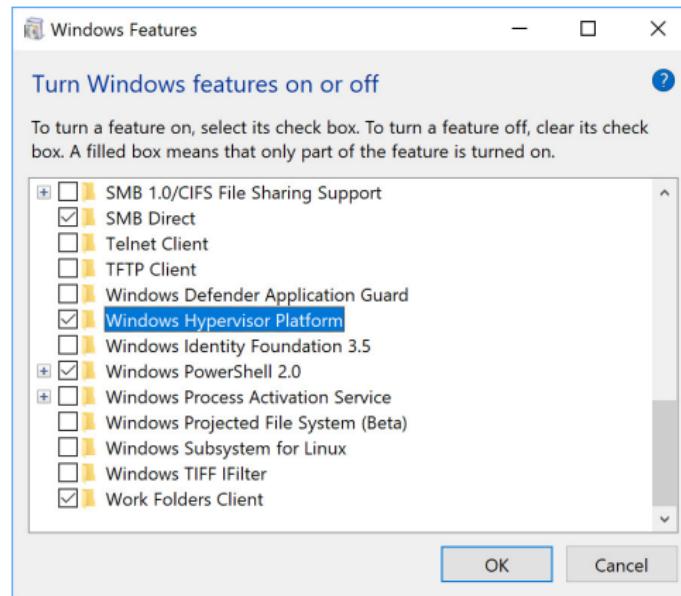
VirtualBox

OpenFOAM

Compile

Test

Control panel → Programs → Turn Windows Features On or Off → Enable the “Windows Hypervisor Platform” → OK. Reboot



This option might be present only in Windows 10 HOME EDITION



# What if I have Windows Home Edition?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

If you try installing docker and you have Windows Home Edition, you might encounter this problem:



The problem is related to the fact that the native hypervisor Hyper-v (Windows Server Virtualization) is not available on **Windows 10 Home**. The hypervisor must be then replaced by an embedded virtual machine.

How can you solve it? **FORUMS/GROUPS on the internet!**

check this link (click on it!): [https://stackoverflow.com/questions/44406422/...](https://stackoverflow.com/questions/44406422/)



# What if I have Windows Home Edition?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

- If you have a 64-bit system and you have enabled virtualization, then you can install docker-toolbox:

<https://docs.docker.com/toolbox/overview/>

- If you have a 32-bit system, then you can:
  - install docker-toolbox + VirtualBox

**NOTE:** if you have tried to install docker-toolbox + VirtualBox and you still get the error "*This computer doesn't have VT-X/AMD-v enabled. Enabling it in the BIOS is mandatory*" (despite you have done it...):

- open a shell
- disable the support to Hyper-V (it is not needed, since you have VirtualBox installed!). Please type:

```
dism.exe/Online /Disable-Feature:Microsoft-Hyper-V  
dism.exe /Online /Enable-Feature:Microsoft-Hyper-V /All
```

More info at this [\[link\]](#).



# How to install the CTTP-PoliMi Docker?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

There are some steps you must follow:

- 1) be sure to have a [Docker-engine](#) installed on your machine:
  - Linux: follow **steps 1 to 3** from:  
<https://openfoam.org/download/7-linux>
  - MacOS: follow **steps 1 + 2** from:  
<https://openfoam.org/download/7-macos>
  - **Windows 10 Professional:** install [Docker Community Edition for Windows](#) at this [link](#); open a Windows PowerShell and from there you should follow the instructions of point 2...
  - Windows 10 Home Edition (64-bit only): you need to install [Docker Desktop](#) [[link](#)] first (see next slides) and THEN go to point 2. In this case, you might need to work a lot on your system to successfully install Docker.
  - Windows 8 or 8.1 (64-bit only): you need to install [Docker Desktop](#).



# Load fpisk/CTTP docker

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

The screenshot shows the Docker Hub interface for the fpisk/cttp repository. At the top, there's a search bar and navigation links for Explore, Repositories, Organizations, Get Help, and a dropdown for 'fpisk'. Below the header, the repository card for 'fpisk/cttp' is displayed. It features a blue cube icon, the repository name 'fpisk/cttp' with a star icon, and a brief description: 'Computational Techniques for Thermochemical Propulsion. Federico Piscaglia, Politecnico di Milano'. It also indicates it's a 'Container'. On the right side of the card, there's a 'Manage Repository' button and a 'Pulls 206' metric with a download icon. The main content area has tabs for 'Overview' (which is selected) and 'Tags'. The 'Overview' tab contains a detailed description of the Docker image, mentioning Ubuntu-18.04, OpenFOAM-6 compilation, and supplementary material for a class on Thermochemical Propulsion. It also credits Prof. Federico Piscaglia from the Dept. of Aerospace Science and Technology, Politecnico di Milano. To the right of this text block is a 'Docker Pull Command' section with a code block containing 'docker pull fpisk/cttp' and a copy icon. Below that is an 'Owner' section showing a profile picture of a person with glasses and the name 'fpisk'.



# How to install the CTTP-PoliMi Docker?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

- 2) from the bash shell (Linux) or from the Windows PowerShell (Windows) or from the Terminal (MacOS), please:

- login to docker:

```
docker login
```

- download and install the docker image of the class:

```
docker pull fpisk/cttp
```

You will start downloading the image of the docker package. **Please note that you are downloading a 3 GB file!**



# How to install the CTTP-PoliMi Docker?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

## Important note for Linux systems:

If you want to use docker, you must add your \$USER to the group docker. This is needed because your user by default does not have enough rights to handle the docker image (like the CTTP-PoliMi container).

So, after the image has been downloaded (point 2), please:

### 3) add **docker group**:

```
sudo groupadd docker
```

### 4) add the user to the group docker

```
sudo usermod -a -G docker $USER
```

### 5) **Log out and log back in** so that your group membership is re-evaluated.



# How to load the CTTP-PoliMi Docker?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

After pulling the Docker image, from your shell you can:

1. list the images available in memory:

```
docker images      or      docker image ls
```

2. share a folder with the native OS:

```
~$ systemFolder=<path-of-your-dataFolder> (in the HOST OS)  
~$ dockerFolder=/home/student/OpenFOAM/student-7/run  
~$ imageID=ad2a91b7a5a1 (check by 'docker image ls' the imageID!)  
~$ docker run --rm -it -v $systemFolder:$dockerFolder $imageID
```

**Note:** if you have Win10 and Docker ToolBox you have to be careful to define the systemFolder path (→ see slide!!).



# Load CTTP-PoliMi docker in Win10-Home Ed.

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

**Note:** if you have Win10 and Docker ToolBox you have to be careful to define the systemFolder path.

Example:

We assume that the User “fede” will share with the docker machine a folder CFD in his folder “Documents” in Windows machine.

1. load Docker ToolBox
2. Open the Windows Powershell in Windows.
3. From the Powershell, please type on one single line:

```
docker run --rm -it
-v //c/Users/<myUserName>/Documents/CFD:/home/student/OpenFOAM/student-7/run
<imageID>
```

**Note:** the shared folder in the OS (Windows) MUST be in a place where you have writing permission as user (e.g. Documents). Do not use “//c/” or any system folder, otherwise the command will not work. In particular, the docker machine will be loaded but the folder will not be mounted.



# Where can I find my dockerID?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

```
fede@macBook:~$ docker image ls
REPOSITORY          TAG      IMAGE ID            CREATED             SIZE
fpisk/cttp          latest   bcbce62cff97    7 weeks ago        4.3GB

fede@macBook:~$
```

## EXAMPLE:

To find your \$dockerID, you must type “`docker image ls`”. In the example, the imageID is `bcbce62cff97`. You are then ready to type:

```
$ dockerFolder=/home/student/OpenFOAM/student-7/run
$ docker run --rm -it -v //c/CFD:$dockerFolder bcbce62cff97
```



# How to test my Docker install?

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

If your Docker installation is successfull, you should be able to get:

```
fede — docker
Last login: Fri Sep 20 18:51:33 on ttys003
[fede@macBook:~$ docker images
REPOSITORY          TAG      IMAGE ID      CREATED       SIZE
fpisk/ctp           latest   bccbed2cff97   8 weeks ago   4.36B
kitematic/hello-world-nginx  latest   03b4557ad7b9   4 years ago  7.91MB
[fede@macBook:~$ loadDocker-7

Welcome to the CFD class!
Computational Techniques for Thermochemical Propulsion
Cod. 81176 - Master of Science in Aeronautical Engineering
POLITECNICO DI MILANO
Prof. Federico Piscaglia
Dept. of Aerospace Science and Technology (DAER)

This course will provide students with an introduction to numerical methods
and analysis techniques used in computational fluid dynamics (CFD) involving
turbulent reacting flows, including the basic approaches and models commonly
used in the literature. The open-source OpenFOAM CFD code
(https://openfoam.org/release/7) has been selected as base tool for the class.

More information about the class is available at
https://piscaglia.aero.polimi.it/cfd/

student@docker-PoliMi:/home/student
```

To test that the folder sharing between Docker and your OS System was successful:

- 1) from the Docker machine

```
$ run (e.g. cd FOAM_RUN)
$ touch testFile
```

- 2) in your OS system (Win, MacOS, Linux), please check in the shared folder if testFile has been generated.



# Official distribution of OpenFOAM in Docker

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

There is a Docker installation of OpenFOAM also provided by the **OpenFOAM Foundation**. More information can be found on the OpenFOAM website:

[openfoam.org/download](http://openfoam.org/download)



# Some commands in Docker

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

- 1) To build an image (at the same level of Dockerfile):

```
docker system prune -a → clean the docker images from memory
```

```
docker build -tag <imageName> .
```

- 2) clean the docker images from memory: docker system prune -a

- 3) to list the images: docker images or docker image ls

- 4) to SAVE the image

```
docker save name > name.tar
```

```
or: docker save -o imageName.tar <imageName>
```

- 5) to load an image from file: docker load -i imageName.tar



Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

# Ubuntu bash shell in Win10



# Using Windows bash

Install

liveDistro

docker

system req.

install

Linux/Mac

win+bash

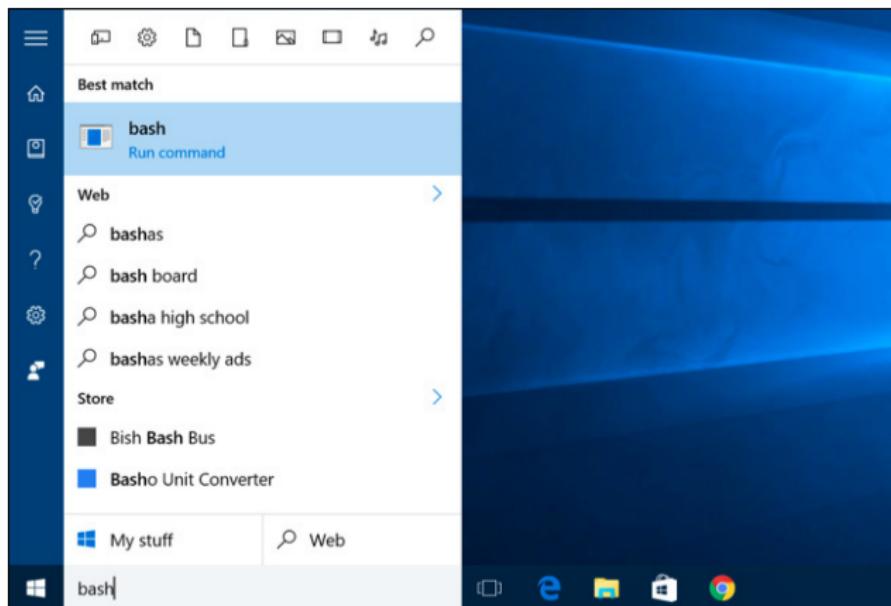
VirtualBox

OpenFOAM

Compile

Test

Open the shell (press the start button and type ‘bash’)





# Using Windows bash

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox

OpenFOAM  
Compile  
Test

- Accept terms of service
- Your bash shell is set. Remember:
  - You are **root**
  - Main filesystem is mounted as `/mnt/c`

```
C:\Windows\System32\bash.exe
-- Beta feature --
This will install Ubuntu on Windows, distributed by Canonical
and licensed under its terms available here:
https://aka.ms/uowterms

Type "y" to continue: y
Downloading from the Windows Store... 100%
Extracting filesystem, this will take a few minutes...
Installation successful! The environment will start momentarily...
root@localhost:/mnt/c/WINDOWS/system32#
```



# Windows 10 Home

Install

liveDistro

docker

system req.

install

Linux/Mac

win+bash

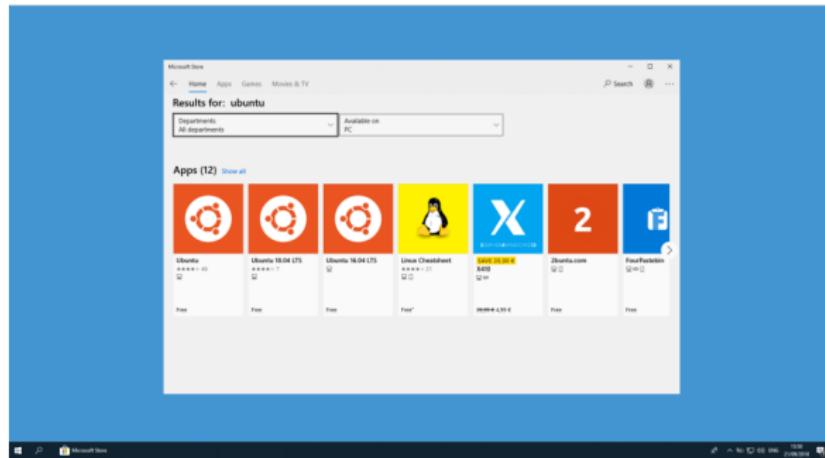
VirtualBox

OpenFOAM

Compile

Test

From Windows 10 Home Edition, you might need to install manually Ubuntu 18.04 LTS from Microsoft store:



Once you have done it, from the bash shell please type:

- sudo apt-get update
- sudo apt-get upgrade
- follow the steps at this [link]



Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

# VirtualBox

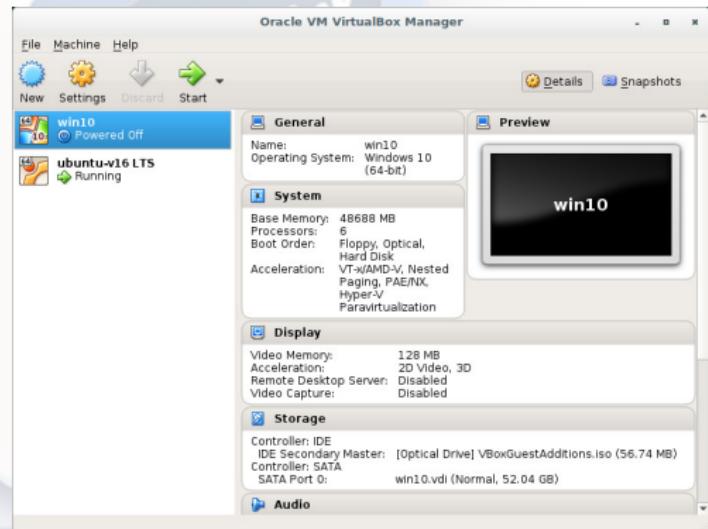


# VirtualBox

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

VirtualBox® is an open-source software for execution of **virtual machines**.

- A Virtual Machine is a virtual OS running upon the OS installed on the physical machine. Examples: Win on Linux, Linux on Win, Linux on Linux, Linux on MacOS, Windows on MacOS.
- the OS installed on the VM is called **GUEST OS**, while OS of the host machine is called **HOST OS**.





# Installing VirtualBox

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

1. Download the latest release of VirtualBox for your *host* OS from [www.virtualbox.org](http://www.virtualbox.org) and install it
2. Download the extension pack from the website
3. Open VirtualBox and install the extension pack:  
**file → preferences → extension → **
4. Download the virtual machine file using the link provided (filesender)
5. Import the VM into your VirtualBox:  
**file → import appliance → browse and select the .ova file**  
Check “reinitialize the MAC address of all network cards”
6. Review the VM settings before starting it.

**Note:** the VM machine settings determine the hardware resources ‘seen’ by the guest OS once it is started. So, if you select 1 CPU for the guest OS, the Virtual Machine will use only one physical core, no matter how many are there on the physical machine



# VirtualBox settings

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

Appliance to import  
/Users/fede/Documents/ubuntu-v16LTS.ova

Appliance settings

Description	Configuration
RAM	16384 MB
DVD	<input checked="" type="checkbox"/>
USB Controller	<input checked="" type="checkbox"/>
Sound Card	<input checked="" type="checkbox"/> ICH AC97
Network Adapter	<input checked="" type="checkbox"/> Intel PRO/1000 MT Desktop (82540EM)
Storage Controller (IDE)	PIIX4
Storage Controller (IDE)	PIIX4
Storage Controller (SATA)	AHCI
Virtual Disk Image	/Users/fede/VirtualBox VMs/ubuntu-v16 LTS_1/ubuntu-...

Reinitialize the MAC address of all network cards

Guided Mode Restore Defaults Go Back Import Cancel

**IMPORTANT NOTE:** check of “reinitialize the MAC address of all network cards” can be done ONLY when you are importing your appliance (→ when you select the .ova file)!



# Guidelines for VM settings

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

- System
  - Memory: no more than half of the global available memory
  - Number of CPU: no more than half of the total number
  - Enable PAE/NX
  - Enable VT-x/AMD-V (**see note 1 and 2**)
- Display
  - Video Memory: maximum available
  - Enable 3D acceleration (**see note 3 and 4**)
- USB
  - ~~Enable USB controller (version 2: see note 3)~~
  - ~~Add a new filter~~
  - Disable USB
- Shared folders
  - Add a shared folder (**see note 3**)
  - Automount, Permanent



# VBox Guest Additions and Shared Folders

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

**Note:** VirtualBox guest additions are already installed in the provided VM. These notes are only for future reference.

To install VBox Guest Additions on the Guest OS:

```
~ $ sudo su
~ # sudo apt-get install -y virtualbox-guest-additions-iso
~ # sudo mount -o loop \
      /usr/share/virtualbox/VBoxGuestAdditions.iso /mnt
~ # cd /mnt
mnt # ./vBoxLinuxAdditions.run
```

---

## Shared folders

- With the VM switched off, go to:  
**Settings → shared folders → Add shared folder**
- **Folder path:** the path to the folder you want to share *on the HOST OS*
- **Folder name:** an identifier for the shared folder in the GUEST OS
- Enable auto-mount
- You can access the shared folder in the guest OS at `/media/sf_<folder name>`



# Some notes on the settings in VirtualBox

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

## VM settings for graphics

- VT-x allows a significant guest speedup for 64-bit hosts. It must be enabled in the BIOS, so it might require a full system reboot and administrative rights.
- Hyper-v is not available on Windows Home. Requires 'VBox Guest additions' to be installed in the Guest OS. These options can be activated only after a first run of the guest OS.
- with some specific hardware (especially laptops) the **OpenGL rendering in the VM is not fully supported**. In case you have trouble in running 3D applications (e.g. ParaView), disable 3D acceleration.

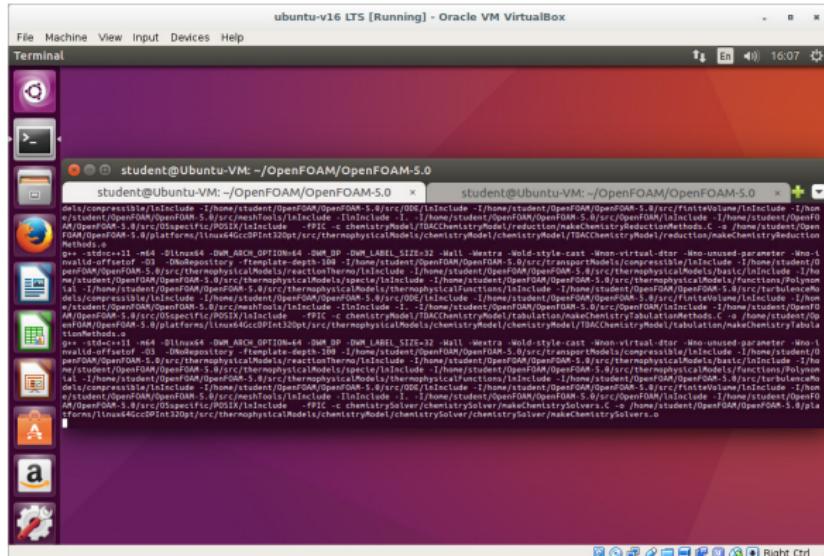


# First run

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox

OpenFOAM  
Compile  
Test

1. Double click on the VM to start it
2. insert password: openFOAM
3. If needed, install Guest Additions (see previous slide)
4. Shutdown the VM and review the advanced options of the previous slide
5. Start again the VM
6. You are all set: use R-CTRL+F to go in fullscreen mode (and back).





Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

# Prerequisites to install OpenFOAM



# Steps to install OpenFOAM

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

**In this class, we will consider installing the latest release of OpenFOAM (currently 7) from the source code**

From now on, we will assume that you have a working environment (either: Linux host, VM, Win10 bash).

The installation of OpenFOAM from source unfolds along the following steps:

1. Install additional packages
2. Choose where to install OpenFOAM (local/shared)
3. Download Third-Party source
4. Download OpenFOAM source
5. Load OpenFOAM in the environment
6. Compile ParaView
7. Compile OpenFOAM
8. Finalize
9. Test



# 1. Install additional packages

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

To successfully compile OpenFOAM and ParaView, you need to have the following packages pre-installed on your Linux distribution:

- GCC compiler  $\geq 4.5$
- FLEX (Fast LEXical analyzer)
- cmake
- QT4 libraries
- GIT version control
- OpenMPI



# 1. Install additional packages

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac

win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

To install the required packages simply run the following commands from the prompt (for Debian/Ubuntu-based installations):

```
~$ sudo apt-get install build-essential flex bison  
cmake zlib1g-dev libboost-system-dev libboost-thread-  
dev libopenmpi-dev openmpi-bin gnuplot libxt-dev  
libreadline-dev libncurses-dev gnuplot gnuplot-x11  
  
~$ sudo apt-get install curl qt4-dev-tools libqt4-dev  
libqt4-opengl-dev freeglut3-dev libqtwebkit-dev  
  
~$ sudo apt-get install libscotch-dev libcgal-dev
```

See also:

<https://www.openfoam.com/documentation/system-requirements.php>



Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

# Steps to install OpenFOAM



## 2. Installation location

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

OpenFOAM can be installed for a single user (local installation), or for many users (shared installation):

- **Local installation:** OpenFOAM is installed in

`$HOME/OpenFOAM/OpenFOAM-7`

- ✓ Each user will 'own' his/her own installation and may update it any time
- ✓ Changes to the code do not affect other users
- ✗ Extra disk space required
- ✗ All users must know how to install OF

- **Shared installation:** OpenFOAM is installed in a folder readable by all users, e.g.:

`/opt/OpenFOAM/OpenFOAM-7`

- ✓ All users will have exactly the same version
- ✓ IT staff is responsible to keep OpenFOAM up to date
- ✓ No need to know how to compile OF
- ✗ Cannot change the code
- ✗ You must be nice with the Sysadmin

In this presentation, we will see how to perform a local installation.



### 3. Download 3rd-party and source

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

“ThirdParty” contains any package which is neither part of OpenFOAM, nor is normally installed on Linux. Nonetheless, they are indispensable to use OpenFOAM (e.g. ParaView)

1. `-$ cd`
2. `~$ mkdir -p $HOME/OpenFOAM`
3. `~$ cd $HOME/OpenFOAM`
4. `$ wget -O - http://dl.openfoam.org/third-party/7 | tar xvz`  
`mv ThirdParty-7-version-7 ThirdParty-7`

“OpenFOAM” will (of course!) contain OpenFOAM source tree, binaries, config files and scripts.

1. `wget -O - http://dl.openfoam.org/source/7 | tar xvz`
2. `mv OpenFOAM-7-version-7 $HOME/OpenFOAM/OpenFOAM-7`



## 4. Load OpenFOAM in the environment

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile

Test

- OpenFOAM relies on a certain number of **System variables** and **aliases**. They are used:
  - To configure system's \$PATH (list of folders the system looks in when searching for an executable)
  - To configure \$LD\_LIBRARY\_PATH (list of folders used to search for dynamic linked libraries)
  - To ease user's life, by avoid typing long strings
- To set all system variables and aliases, the user must **source** (i.e. execute) the file ~/OpenFOAM/OpenFOAM-7/etc/bashrc:

```
source /OpenFOAM/OpenFOAM-7/etc/bashrc
```
- System variables are bound to the shell. Every time a new shell is opened, bashrc must be sourced. To do it automatically:
  1. open the file \$HOME/.bashrc by a text editor (gedit, emacs, vi);
  2. add the source command to the bottom of the file
- try echo \$PATH **before and after sourcing** the bashrc
- the command alias shows all defined aliases (try!)



## 5. Compile the code

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

- Start the compilation by running:

```
~$ foam
```

```
OpenFOAM-7 $ ./Allwmake [-j [<nProcs>]] > log.allwmake >&1
```

- Option `[-j [nProcs]]` allows multithreaded compilation. If `nProcs` is not specified, all available cores will be used
- Both compiler output and errors are saved on `log.allwmake`



# Final steps

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

- The final step is to create a folder for your cases

- By default, OpenFOAM set a system variable:

```
FOAM_RUN = $HOME/OpenFOAM/$USER-$WM_PROJECT_VERSION/run
```

- To see the actual value of \$FOAM\_RUN, type: echo \$FOAM\_RUN

- The alias run expands to cd \$FOAM\_RUN

- Right now, \$FOAM\_RUN does not exist. To create it, type:

```
mkdir -p $FOAM_RUN
```

- Then you can cd to it: run



## Final steps (win-bash only)

Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

- You can use the pre-compiled version of ParaView for Windows...

<https://www.paraview.org/download/>

- ...but you must have a shared folder between bash and Windows:

1. Create a folder in windows on the main drive, e.g.:

C:\OpenFOAM\cases

2. Open the bash shell and go into the run folder:

run

3. Create a link to the windows folder:

ln -s /mnt/c/OpenFOAM/cases .

- Files and folders in C:\OpenFOAM\cases are seen in bash and vice-versa

- **Please use Linux conventions for filenames (case-sensitive, no spaces, no accents and special characters but { . - \_ } )**



Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

# Test your installation



# Your first OpenFOAM case

Install

liveDistro

docker

system req.

install

Linux/Mac

win+bash

VirtualBox

OpenFOAM

Compile

Test

1. Close all terminals and open a new one
2. Go to the ‘run’ folder: `run`
3. Make a working copy of the tutorial folder: `cp -r $FOAM_TUTORIALS .`
4. Change dir: `cd incompressible/icoFoam/cavity/cavity`
5. Generate the mesh: `blockMesh`
6. run solver: `icoFoam`
7. check the results: `paraFoam`



Install  
liveDistro  
docker  
system req.  
install  
Linux/Mac  
win+bash  
VirtualBox  
OpenFOAM  
Compile  
Test

# Thank you for your attention!

contact: [federico.piscaglia@polimi.it](mailto:federico.piscaglia@polimi.it)