Introduction to open-source CFD

Open-Source CFD Course 2021 – Lecture 1

Miklós BALOGH and Josh DAVIDSON

2021

Table of Contents

About the course

- Main objective: Getting familiar with Open-Source CFD
- Outcomes:
  - Experience with Linux OS and its command line environment
  - Ability to solve practical problems using OpenFOAM and other open-source tools
- Lectures and laboratory:
  - Theoretical part (every second week): getting familiar with a given topic
  - Laboratory part (every week): deepen your knowledge via practicing
  - Assignment part (after lab.): check your knowledge with answering to questions
  - Optional, but strongly suggested part (at home): further intensification with extra practicing

Course outline: Assessment

- Lectures (6×2×45 mins. online)
- Practical (12×2×45 mins. individual)
- Individual projects (announcements will be available soon)
  - Consultation
  - Report submission (at 13th week)
  - Final report submission (at 14th week)
- Final grade
  - Mid term exam on 11th week (retake 12th week): 50%
  - Individual project report: 50%
  - Additional bonus marks: maximum extra 15%

Course Outline: Covered theoretical topics

- 1. week: Introduction to OpenFOAM and Linux (MB)
- 3. week: Structure of OpenFOAM, simple problems (JD)
- 5. week: Open-Source Meshing (MB)
- 7. week: Solvers and boundary conditions (JD)
- 9. week: Advanced post-processing (MB)
- 11 week: Summary, Tips and Tricks (JD & MB)
- 13 week: Consultation (JD & MB)

Course Outline: Examination (on MS Teams)

- 1-10. week: Laboratory sessions with assignments for bonus marks
- 11. week: Mid-term exam (similar problems than at the assignment sessions)
- 4-14. week: Individual projects
  - 3. week: proposals (own idea or project picked from a list)
  - 5-13. week: elaboration individually and consultations at lab sessions
  - 13. week: project report submission (4-6 page long report)
  - 14. week: final report submission (after review)

About OpenFOAM - What is OpenFOAM

- **Open** source **F**ield **O**peration **A**nd **M**anipulation
- It is free and open-source, distributed under the General Public License (GPL).
- The GPL gives users the freedom to modify and redistribute the software and a guarantee of continued free use, within the terms of the licence.

About OpenFOAM - History

- FOAM was created by Henry Weller in 1989.
- Released as OpenFOAM in 2004, while OpenCFD is founded.
- OpenCFD was acquired by SGI Corp in 2011 than they sell OpenCFD Ltd. to ESI group in 2013.
- Therefore the project co-founders formed the OpenFOAM Foundation. This not-for-profit organisation responsible for the guardianship of OpenFOAM and its GPL distribution.
- The latest versions (foundation, ".org" version 8.0 and ESI Group ".com" version 20.12) used in this course are released in Dec. 2020.
- Other versions: OpenFOAM-dev (rolling release) and foam-extend (version 4.1)

OpenFOAM simulation case directory tree

- *<case>*
  - *system*
    - *controlDict*
    - *fvSchemes*
    - *fvSolution*
  - *constant*
    - *...Properties*
    - *polyMesh*
      - *points*
      - *faces*
      - *owner*
      - *neighbour*
      - *boundary*
  - time directories

Equation:

$$\frac{\partial\left(\rho\mathbf{U}\right)}{\partial t}+\nabla\cdot\left(\rho\mathbf{U}\otimes\mathbf{U}\right)-\nabla\cdot\left(\mu\nabla\mathbf{U}\right)=-\nabla p$$

High-level `C++` syntax of the solver:

```
solve
(
   fvm::ddt(rho, U)
 + fvm::div(phi, U)
 + laplacian(mu,U)
   ==
 - fvc::grad(p)
);
```

OpenFOAM case syntax

- Settings are also stored in `C++` syntax form.
  - Simulation control: case/system/controlDict
  - Solution control: case/system/fvSolution
  - Numerical schemes: case/system/fvSchemes

```
// e.g. divergence schemes in fvSchemes
divSchemes
{
    default     none;
    div(phi,U) bounded Gauss linearUpwind grad(U);
}
```

About Linux

- Unix-like computer operating system
- Developed by the community (free and open-source model for development and distribution)
- Initial release: September 17, 1991 by Linus Torvalds
- Free Linux distributions: Arch Linux, CentOS, Debian, Fedora, Gentoo Linux, Linux Mint, Mageia, openSUSE and Ubuntu
- Commercial Linux distros: Red Hat Enterprise Linux, SUSE Linux Enterprise Server
- Note: From smartphones to cars, supercomputers and home appliances, the Linux operating system is everywhere

Structure of Linux

- **Bootloader:** manages the boot process
- **Kernel:** core of the system which manages the CPU, memory, and peripheral devices.
- **Daemons:** background services (printing, sound, scheduling, etc).
- **Shell:** Linux command line environment, that allows you to control the computer via commands.
- **Graphical Server:** sub-system that displays the graphics on your monitor. Referred as X server.
- **Desktop Environment:** The GUI (Unity, GNOME, Cinnamon, Enlightenment, KDE, XFCE, LXDE, LXQt).
- **Applications:** Programs for different purposes.

The Shell: Linux command line

- Often called as **terminal** under desktop environments.
- This is a program that takes your commands from the keyboard and gives them to the operating system to perform.
- Most of the apps and their settings can be accessed via commands.
- Terminal is the only option to run simulations on HPC clusters and supercomputers.
- You should have a **3-button mouse** if you want to use Linux effectively.
- You should use the **tab button** frequently in file system navigation.

Basic shell commands

Command	Description
ls	list directory contents
ls -l	list in long format
ls -ltr	list in long format, sort by time, reverse order
ls –help	see other options for ls command
man ls	see manual entries of ls command (to exit press q)
cd ..	change directory to the parent directory
cd /bin	change directory to /bin
cd	change directory to $HOME (home dir of user)
cd /OpenFOAM	change directory to $HOME/OpenFOAM
pwd	print name of current directory
whoami	print the login name of the user
echo "Hello World"	print to the next line "Hello World"
echo $HOME	print the value of $HOME environmental variable
find -name OpenFOAM	find files or directories named OpenFOAM
man find	manual of find command (to exit press q)
which ls	print the path for ls command

Text file related commands

Command	Description
cat file	print file content
less file	print file content and scroll (exit: q)
	searching: type "/sys" to search "sys"
	type "n" for next, "N" for previous
wc file	print document statistic: number of lines, words and bytes
wc -w file	print number of words
man wc	manual entry for wc command
head file	print the first part of file
tail file	print last part of file
grep pattern file	print lines matching the pattern in the file
sort file	sort the lines in the file
sort -r file	sort lines of text, reverse order
tac file	print file content, reverse order
grep -ri pattern dir	find files whose contain pattern in dir (-i: ignore case)
man sed; man awk	manuals of more advanced text processing programs

File system related commands

Command	Description
cp file1 file2	copy file1 to file2
mv file1 file2	rename file1 to file2
mv file dir/	move file to dir
rm file	remove file
mkdir dir	make directory "dir"
rm -r dir	remove directory
sort file1 > file2	sort the content of file1 to file2 (overwriting file2)
sort < file1 > file2	sort file1 to file2
sort < file1 >> file2	sort file1 and add to file2
chmod +x file	change file permissions to executable for all
chmod +rwx file	change file permissions to readable, writable, executable for all
chmod u+rw file	change file permissions to read and writable for the current user

Process control and performance monitoring commands

Command	Description
ps	list running processes
ps aux | less	list all processes
ps aux | grep pattern	list processes which contain pattern
top	interactive realtime performance list of processes
kill 20123 -9	kill/terminate process immediately with ID=20123
free -h	report free memory
df -h	report disk space usage
du -sh	estimate file space usage

Useful tricks

- Using up and down arrows in terminal:
  - You can browse in the history of executed commands.
- Using your mouse to speed up the work:
  - Selecting a word: move the cursor over the word and double-click
  - Selecting a line: move the cursor over the line and triple-click
  - Paste from clipboard: Push middle button/push scroll wheel/push the two mouse side button at once
- Using tab button to typing faster:
  - When you are typing in terminal, the tab button activates the automatic path completion.
  - If there are more completion option, hit the tab again will list all of them.

Installation on Linux

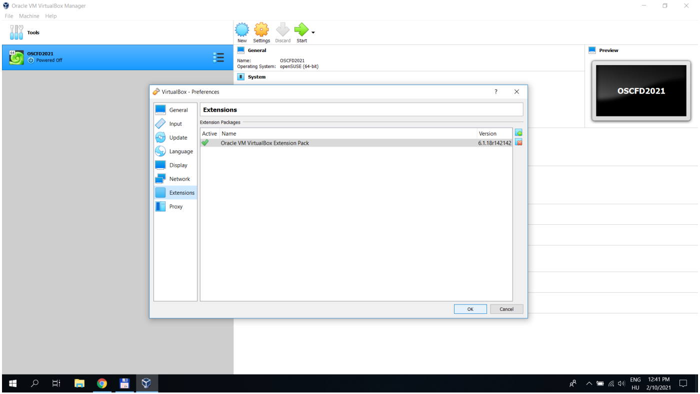
- OpenFOAM is developed for Linux, thus it is fully supported on Unix-like platforms.
- If you have Linux installed, you have several options:
  - Installation from repository using the package manager.
  - Installation via docker cross-platform container. This is available for all operation system.
  - Compilation from source (Only for experts): The source pack distribution can be downloaded from the official web-site or you can use the Git repository. Your compiled code will be optimized for your operation system. This option is available for every Unix-like platform.

Installation on Windows

- Options:
  - Installation via windows installer (using docker cross-platform container).
  - You can use a virtualization platform, e.g. Oracle VirtualBox.
  - Install the Windows Subsystem for Linux and install the Ubuntu 18.04 LTS Linux Distribution on windows. Then the options for Linux can be chosen.

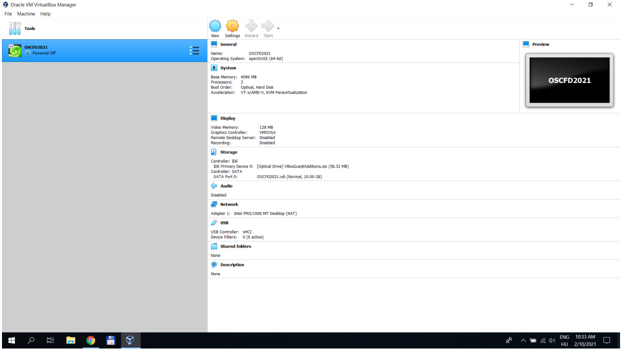
Install VirtualBox

- Download and install the latest version available.
- Download the extensions (same/latest version)
- VirtualBox Manager: File → Preferences → Extensions



Install virtual image

- VirtualBox Manager: New
- Check the option: Use an existing virtual hard-disk file

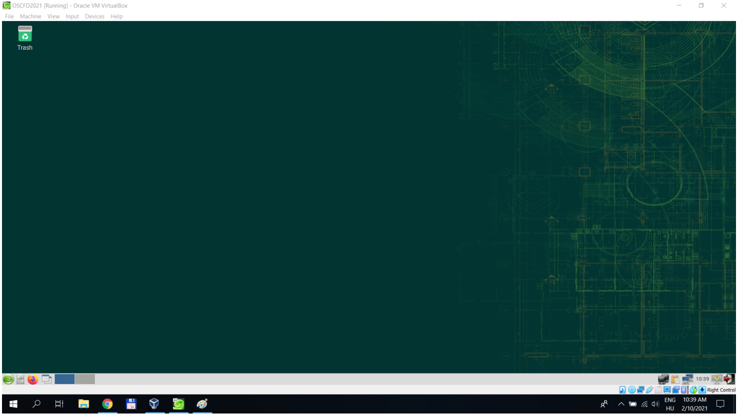


Virtual machine settings

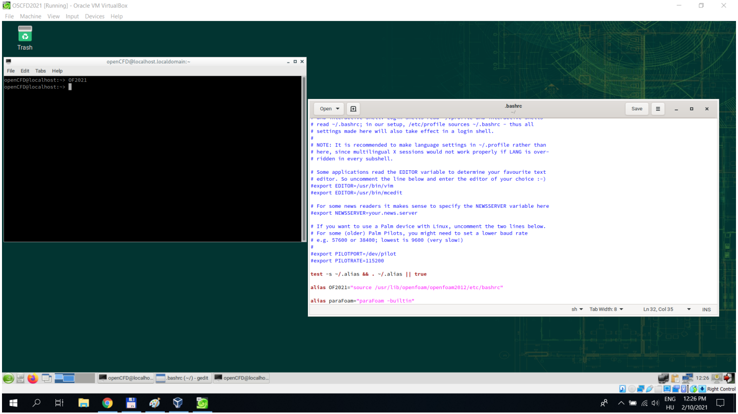
- General → Advanced tab → set both drop-down menu to Bidirectional
- System
  - Set affordable system resources for the guest OS
  - Un-check floppy from the boot order list
  - Check Enable I/O APIC
- Display
  - Un-check 2D and 3D acceleration
- Storage
  - Controller Sata → Check Use Host I/O Cache
  - Shared Folders → Add → Browse
- USB
  - Check USB 3.0 (xHCI) Controller
  - Set affordable display resources for the guest OS

Running the virtual OS

- Start suseLXDE2021 from the VirtualBox Manager
- Login with password: OSCFD2021 (autologin is disabled)



Loading OpenFOAM



Testing OpenFOAM

After OpenFOAM is installed and loaded successfully, several Linux environment variables are available:

Variable	Description
$WM_PROJECT_DIR	OpenFOAM installation directory
$FOAM_TUTORIAL	Tutorial cases
$FOAM_RUN	User run directory

Testing installation with running a case in LXTerminal:

```
ll $FOAM_TUTORIALS # List the tutorials
mkdir -p $FOAM_RUN # Create run folder
cp -r $FOAM_TUTORIALS $FOAM_RUN
cd $FOAM_RUN/tutorials/incompressible
cd icoFoam/cavity/cavity
blockMesh
icoFoam
paraFoam
```

Assignments

1. Install OpenFOAM and its environment
2. Familiarize yourself with the command line environment
   - List the content of your /home directory
   - Create a folder /home/workdir (if it is not there)
   - Change directory to /home/workdir
   - List its content
   - Copy OpenFOAM tutorials to /home/workdir
   - List its content
   - Find folders named "cavity"