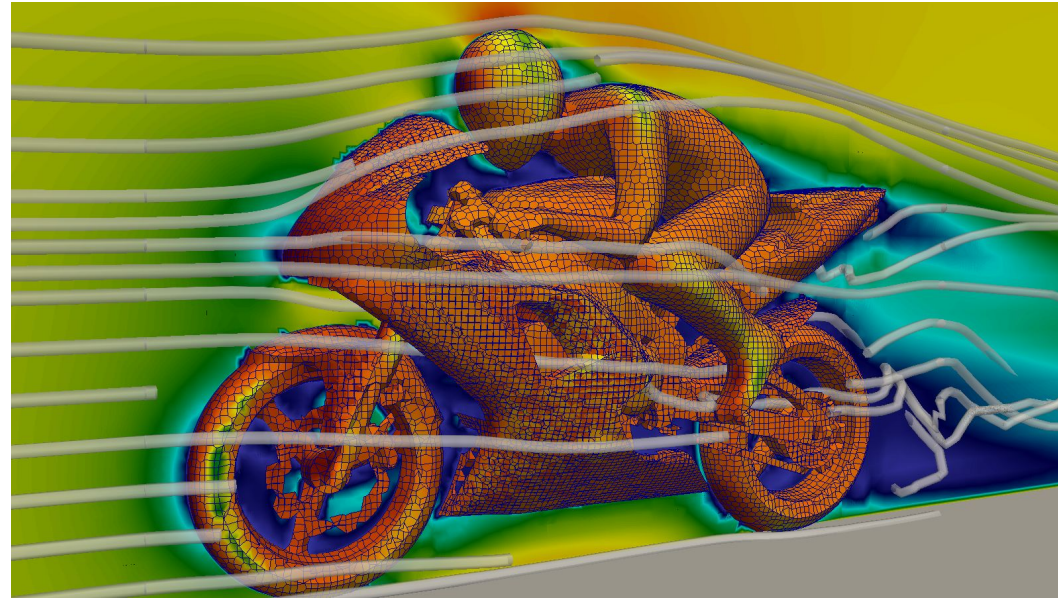


# Open Source Computational Fluid Dynamics



*An MSc course to gain extended knowledge in Computational Fluid Dynamics (CFD) using open source software.*

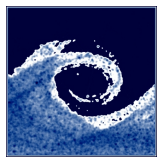
**Teachers:** Miklós Balogh and Zoltán Hernádi

Department of Fluid Mechanics, Budapest University of Technology and Economics



M Ű E G Y E T E M 1 7 8 2

Open Source CFD @ Budapest University of Technology and Economics

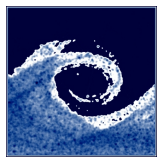


# Course description

- Introduction to OpenFOAM simulations (and linux, gnuplot, paraview).
- Installation on several Linux distributions and virtual systems.
- Solution of simple fluid dynamics problems using OpenFOAM.
- Detailed introduction to OpenFOAM software components:
  - pre-processing (meshing tools and utilities)
  - solving (standard applications, user applications)
  - post-processing (ParaView).
- Single phase stationary and transient flows, turbulence, compressible flows.
- Multiphase and reactive flows.
- Extension of OpenFOAM capabilities by program code development in C++.
- Individual projects using OpenFOAM.
- Further open source CFD tools.

## Course website

<http://www.ara.bme.hu/~hernadi/OpenFOAM>

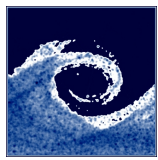


# Planned schedule

- **week 1:** introduction to OpenFOAM, installing
- **week 2:** solving simple fluid flow problems
- **week 3:** studying software components
- **week 4:** stationary and transient flows
- **week 5:** turbulent and compressible flows
- **week 6:** multiphase and reactive flows, **individual project proposals**
- **week 7:** OpenFOAM programming (C++), **individual project declarations**
- **week 8: mid-term exam**
- **week 9-11:** further open source software (e.g. Palabos, Octave, Git)
- **week 12-14: presentations of individual projects**

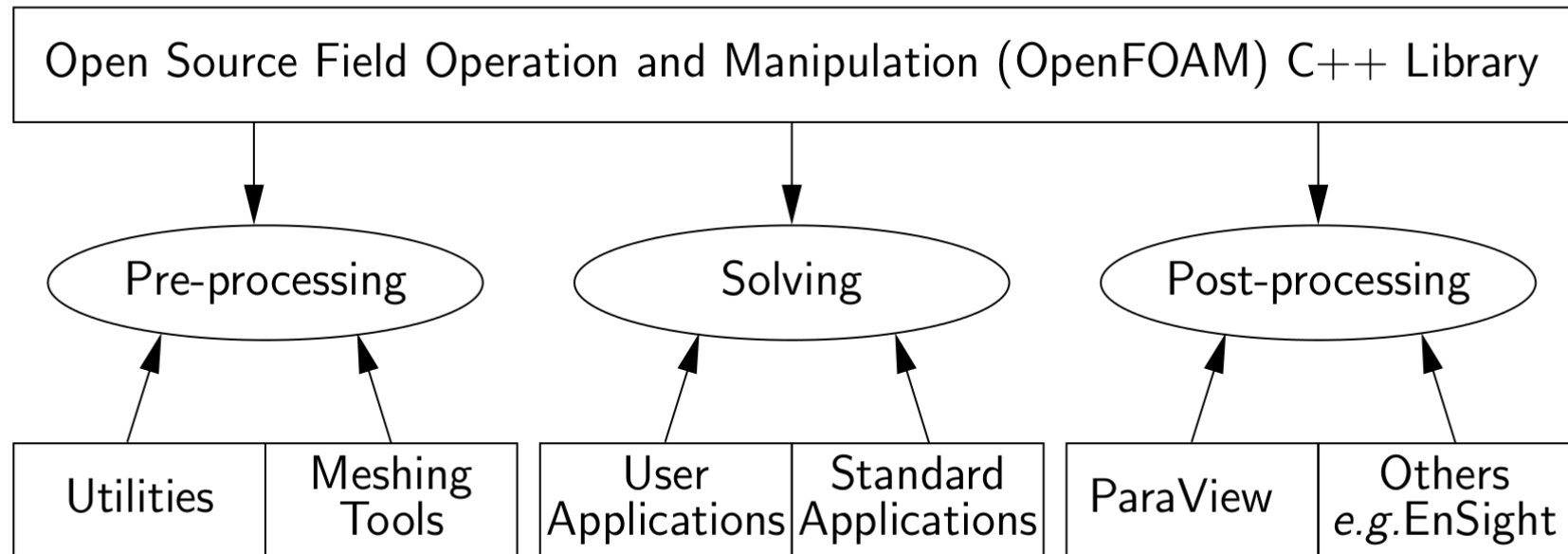
## Final grade

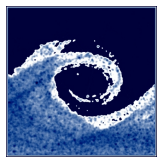
The grading is based on 1 mid-term exam (50% in final grade) and an individual project (50% in final grade). In-class activity will be rewarded by bonus points.



# Introduction to OpenFOAM

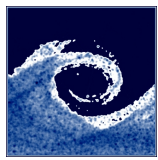
- **Open** source **F**ield **O**peration **A**nd **M**anipulation
- *C++ toolbox*, mainly designed for Computational Fluid Dynamics
- *Linux* is fully supported by OpenFOAM.





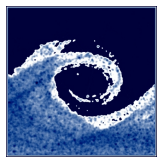
# Linux operating system

- Linux comes in different flavours: **distributions**.
- Most popular Linux distributions are e.g. Arch Linux, CentOS, Debian, **Fedora**, Mageia, Linux Mint, **openSUSE**, PCLinuxOS, Slackware, **Ubuntu**.
- Different distributions have different desktop environments, but the linux **command-line** is very similar.
- The *command-line* can be accessed through a **terminal emulator**, e.g. gnome-terminal, LXTerminal, rxvt, xterm.



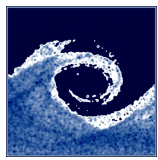
# Commands for navigating in file system

ls	list directory contents
ls -l	(use a long listing format)
ls -ltr	(long listing, sort by time, reverse order)
ls --help	(to see other options)
man ls	manual of ls command (to exit press <code>q</code> )
cd ..	change directory to parent (relative to <code>\$PWD</code> )
cd /bin	change directory to /bin (absolute position)
cd ~/OpenFOAM	change directory to <code>\$HOME/OpenFOAM</code>
pwd	print name of current/working directory
echo HOME	display a line of text: HOME
echo \$HOME	display a variable: <code>\$HOME</code>
cd	change directory to <code>\$HOME</code>
find -name OpenFOAM	find files or directories named OpenFOAM
man find	manual of find command (to exit press <code>q</code> )
which ls	locate a command



# Commands for texts in files

<code>cd \$WM_PROJECT_DIR</code>	change directory to OpenFOAM installation
<code>cat Allwmake</code>	print file content
<code>less Allwmake</code>	print file content and scroll (exit: <code>q</code> ) (type <code>/sys</code> to search "sys") (type <code>n</code> for next, <code>N</code> for previous)
<code>wc Allwmake</code>	print newline, word, and byte counts
<code>wc -w Allwmake</code>	print word counts
<code>man wc</code>	manual for <code>wc</code> command
<code>head Allwmake</code>	output first part of file
<code>tail Allwmake</code>	output last part of file
<code>grep echo Allwmake</code>	print lines matching a pattern
<code>sort Allwmake</code>	sort lines of text
<code>sort -r Allwmake</code>	sort lines of text, reverse order
<code>tac Allwmake</code>	print file content, reverse order
<code>grep -ri piso src</code>	find files with content (-i: ignore case)
<code>sort Allwmake   uniq</code>	omit reported lines in sorted output
<code>man sed; man awk</code>	manuals of more advanced text processing



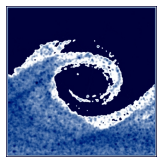
# Commands for manipulating files

<code>cp Allwmake ~/test1</code>	copy file
<code>mv ~/test1 ~/test2</code>	rename file
<code>mv ~/test2 ~/OpenFOAM/</code>	move file
<code>rm ~/OpenFOAM/test2</code>	remove file
<code>mkdir ~/test</code>	make directory
<code>rmdir ~/test</code>	remove empty directory
<code>man rm</code>	manual for rm command (see -r option!)
<code>sort Allwmake &gt;~/test</code>	create file by redirecting
<code>sort Allwmake &gt;&gt;~/test</code>	overwrite file by redirecting
<code>sort &lt;~/test &gt;&gt;~/sorted</code>	sort redirected input and save output
<code>sort &lt;~/test &gt;&gt;&gt;~/sorted</code>	sort redirected input and add output
<code>chmod 000 ~/test</code>	change file mode (check with <code>ls -l</code> )
<code>chmod u+rw ~/test</code>	change file mode (check with <code>ls -l</code> )

## **Note**

Read (4), write (2), execute (1). E.g. `rxr-r-x` = 755, `rw-r--r--` = 644.





# Job control and performance monitoring

ps	list current processes
ps aux   less	list all processes
top	interactive display of processes
kill	kill/terminate process
free -h	report free memory
df -h	report disk space usage
du -sh	estimate file space usage

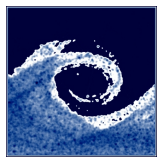
## ***Tip***

Use `TAB` button for auto-completion of commands!

Visit <http://linuxcommand.org> for learning more commands.

## ***Note***

Recommended text editor GUI: `gedit`



# Installing OpenFOAM

- Linux is fully supported by OpenFOAM.
- On non-Linux systems, you can use a virtualization platform, e.g. VirtualBox. You can download a VirtualBox image file from the course website:

<http://www.ara.bme.hu/~hernadi/OpenFOAM/virtualbox.html>

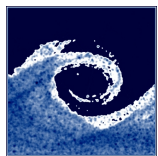
- If you have Linux installed, you can install OpenFOAM from

<http://www.openfoam.org/download/>

- Easy installation is possible on Ubuntu, SuSE or Fedora. For these systems, you don't need to compile OpenFOAM source code.
- Source pack distribution can be downloaded and compiled.
- If you use Git version control system, you can compile OpenFOAM source code with the latest fixes.

## ***Warning***

Compiling OpenFOAM from source usually take several hours! **Be patient!**



# Testing OpenFOAM installation

If OpenFOAM is installed, several Linux environment variables are available, e.g

\$WM_PROJECT_DIR	OpenFOAM installation directory
\$FOAM_RUN	OpenFOAM user directory
\$FOAM_TUTORIALS	OpenFOAM tutorials directory

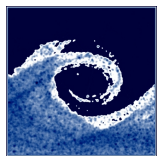
## **Note**

You can list other environment variables by

```
echo $FOAM_<TAB><TAB>  
env | grep 'WM\|FOAM' | less
```

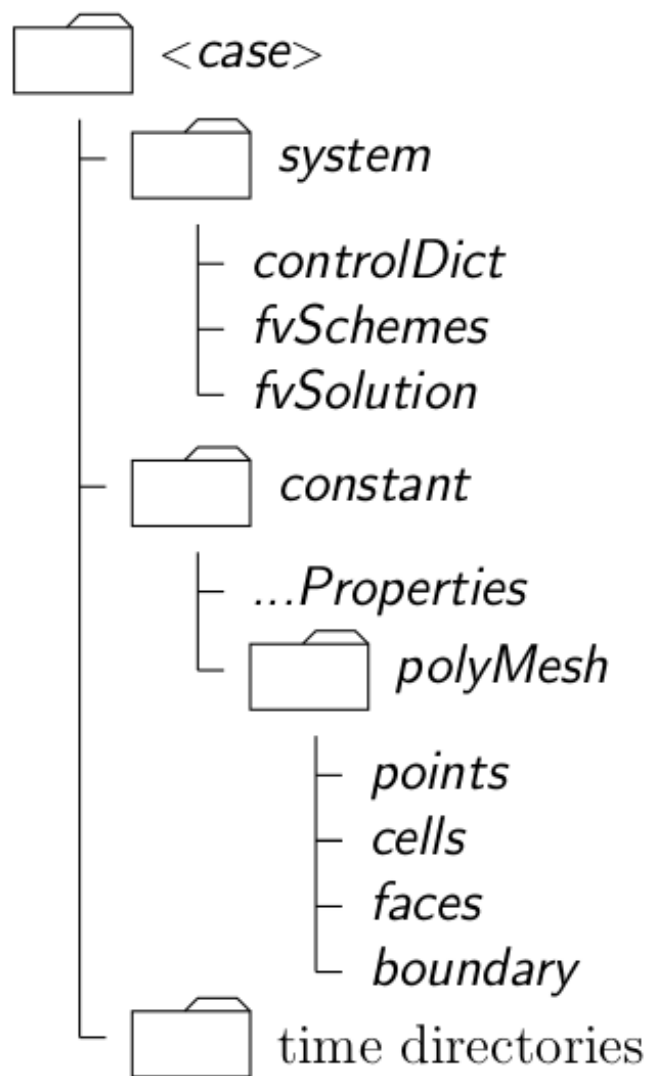
The usual way of testing OpenFOAM is to run a simple simulation:

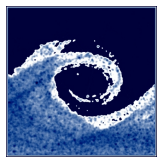
```
mkdir -p $FOAM_RUN  
cp -r $FOAM_TUTORIALS $FOAM_RUN  
cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity  
blockMesh  
icoFoam  
paraFoam
```



# OpenFOAM cases

The basic directory structure:





# How OpenFOAM solvers work

Partial differential equation

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot \phi \mathbf{U} - \nabla \cdot \mu \nabla \mathbf{U} = -\nabla p$$

is represented using high-level C++ syntax inside the solver:

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

Numerical treatment settings (discretization schemes and linear-solvers) are inside OpenFOAM cases (*fvSchemes*, *fvSolution*).

```
divSchemes
{
    default           none;
    div(phi,U)        bounded Gauss linearUpwind grad(U);
}
```