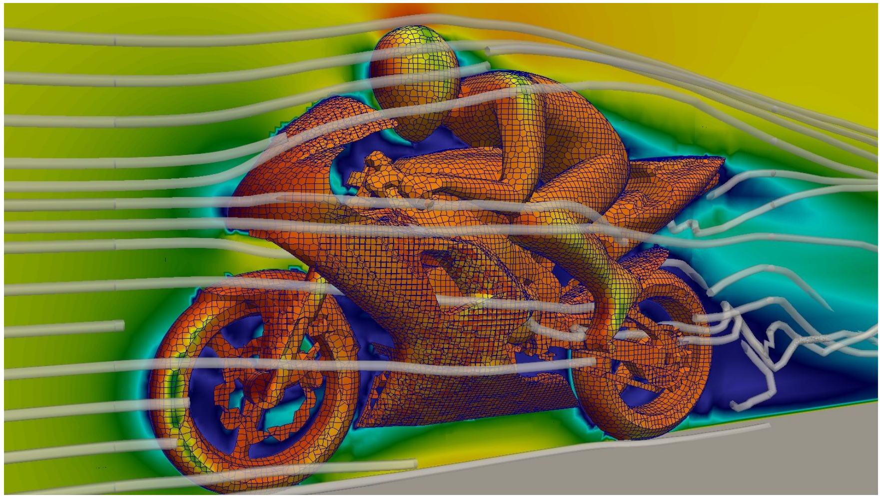
Open Source Computational Fluid Dynamics



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

Zoltán Hernádi

Department of Fluid Mechanics

Budapest University of Technology and Economics

Mesh generation with GMSH

- GMSH: open source (GPLv2+) software for meshing
- Website: http://www.geuz.org/gmsh
- Originally designed for finite element method (FEM)
- OpenFOAM (FVM) can effectively use GMSH meshes
  - 1. In GMSH:
    - Create geometry in GUI or GEO file
    - Define mesh resolution requirements in GEO file
    - Define boundary conditions in GEO file
    - Run: **`gmsh -3 yourGeometry.geo`**
  - 2. In OpenFOAM:
    - Run: **`gmshToFoam yourGeometry.msh`**

Example GEO file

```
lc = 0.05;               // characteristic length
Mesh.VolumeFaces = 1;    // display faces of volume mesh
Mesh.RecombineAll = 1;   // triangulation -> quadrilateral mesh
Mesh.Algorithm = 8;      // Delaunay for quads
```

Parameters

```
Point(1) = {0, 0, 0, lc};
Point(2) = {1, 0, 0, 0.5*lc};
Point(3) = {1.2, 0.2, 0, 0.5*lc};
Point(4) = {2, 0.2, 0, lc};
Point(5) = {2, 0.4, 0, lc};
Point(6) = {-0, 0.4, 0, lc};
```

Points

```
Line(1) = {1, 2};
Line(2) = {2, 3};
Line(3) = {3, 4};
Line(4) = {4, 5};
Line(5) = {5, 6};
Line(6) = {6, 1};
```

Lines

```
Line Loop(7) = {5, 6, 1, 2, 3, 4};
Plane Surface(8) = {7};
```

2D surface

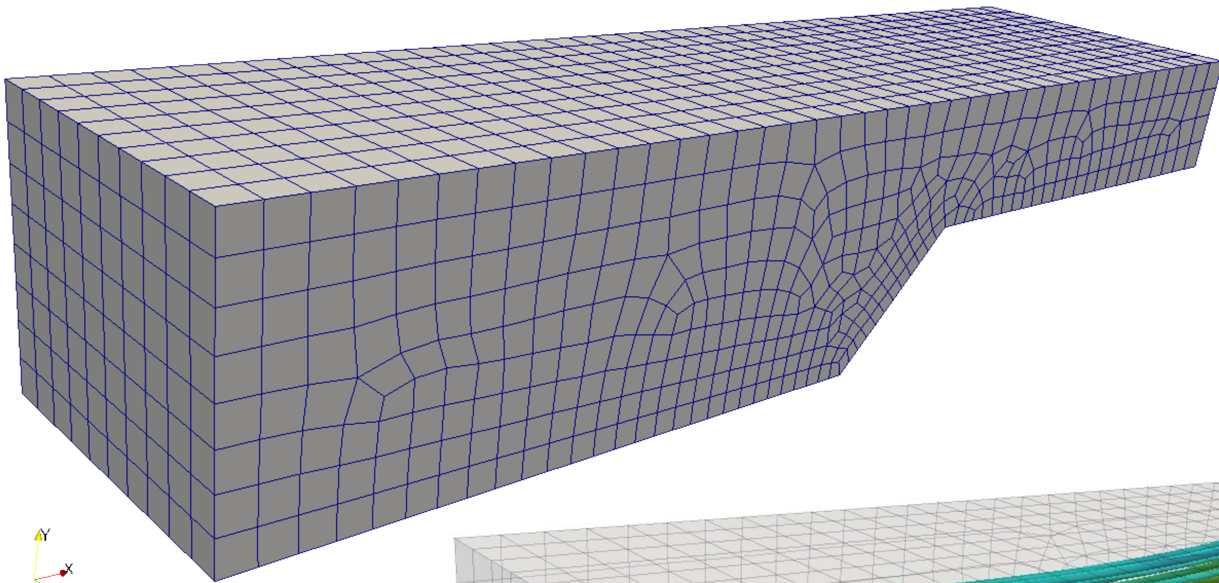
```
Extrude {0, 0, 0.5} {
  Surface{8}; Layers{10}; Recombine;
}
```

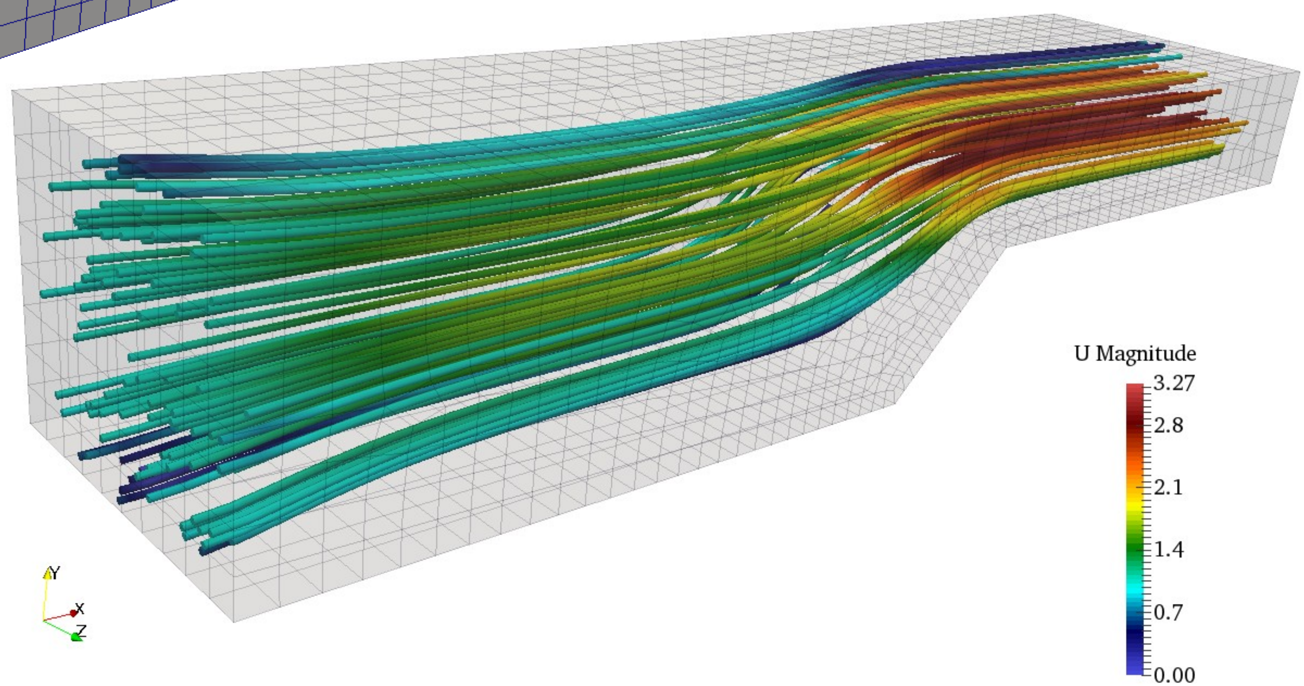
2D extrusion → 3D mesh

```
Physical Surface("inlet") = {23};
Physical Surface("wall") = {27, 31, 35, 40, 19, 8};
Physical Surface("outlet") = {39};
Physical Volume(44) = {1};
```

Boundary conditions
+ volume definition!

Mesh and OpenFOAM flow simulation





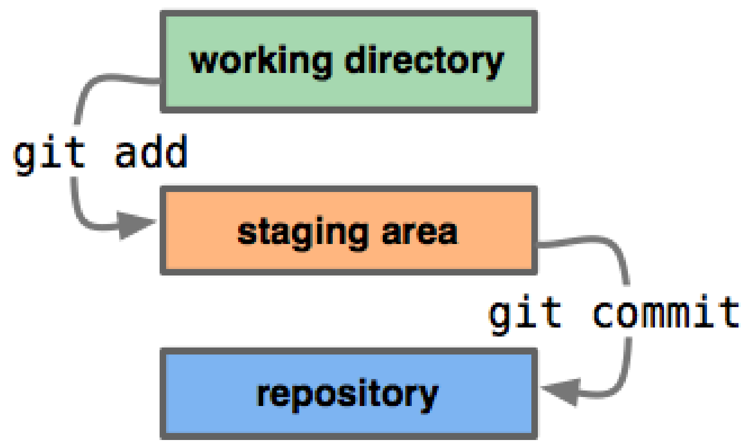
Advanced programming in OpenFOAM

- Revision control: git
- Debugging:
  - *Info* statements
    1. Insert *Info* statements to the source code
    2. Re-compile code
    3. Find and remove bugs
    4. Remove *Info* statements
  - DebugSwitches in $WM_PROJECT_DIR/etc/controlDict

    e.g. modify *lduMatrix* to 2
  - GNU Debugger (GDB)

First git commit

```
# Global configuration of GIT system:
git config --global user.name "Zoltan Hernadi"
git config --global user.email "zhernadi"
git config --global core.editor "gedit"
git config --global color.ui true
git init # initialize repo
echo "My content" > my_file
git add .
git commit -m "Initial commit"
```



Using git

```
git status
git log
git diff           # difference between working directory and stage+repo
git diff --staged # difference between stage and repo
git rm file_to_delete.txt # remove file
git mv old_file.txt new_place/new_file.txt # rename/move
git diff changed_file.txt

# Undo changes in working directory (copy from repo to working directory):
git checkout -- changed_file.txt

# Undo changes in stage:
git reset HEAD changed_file.txt

# Copy old version of file from repo to stage:
git checkout a3076290213d4 -- file_old_version.txt

# Undo full commit by creating a new commit:
git revert c32d0130762

git clean -n # show untracked files to be removed in working directory
git clean -f # remove untracked files in working directory
```

Using git

```
git help
git help log

git log --oneline
git show a3076290213d4
git show HEAD^
git show HEAD~1
git show HEAD^^^
git show HEAD~3

git remote                  # list remotes
git push                    # from local to remote
git fetch                   # from remote to local
git merge origin/master     # after fetch: merge to local
git pull                    # git fetch + git merge
```

OpenFOAM git repository:
https://github.com/OpenFOAM

Debugging with GDB

- Compile OpenFOAM with debug flag:
  `WM_COMPILE_OPTION=Debug wmake`
- either re-compile the complete code of OpenFOAM
- or accept that you can debug the re-compiled code only

GDB commands:

- b: break (set breakpoint)
- r: run (start program)
- n: next (execute next program line)
- c: continue
- l: list nearby source-code lines
- l 12: list source-code lines nearby line 12
- info locals: show local variables
- p var: print variable content
- p var = 1: change variable content to 1
- help name: show information about GDB command name

OpenFOAM information sources

- Main website:
  http://www.openfoam.org
- Unofficial OpenFOAM wiki:
  http://openfoamwiki.net
- CFD Online Forum:
  http://www.cfd-online.com/Forums/openfoam/
- PhD course:
  http://www.tfd.chalmers.se/~hani/kurser/OS_CFD/
- OpenFOAM workshop:
  http://openfoamworkshop.org
- OpenFOAM training:
  http://www.openfoam.com/training/