BURSA TECHNICAL UNIVERSITY

Department of Mechanical Engineering
Computational Fluid Dynamics
MECHT0505

Lecture 1

Asst. Prof. Dr. Levent Aydınbakar

Asst. Prof. Dr. Levent Aydınbakar

- Education
  - ESOGÜ Makine Mühendisliği
  - Cranfield University, UK, MSc in CFD
  - Waseda University, JP, PhD in CFD
- Courses
  - Computer Programming
  - Akışkanlar Mekaniği
  - Introduction to FEM
  - Computational Fluid Dynamics
  - Hesaplamalı Akışkanlar Dinamiği (Grad)
- Research
  - Computational Mechanics

MECHT0505 CFD Course Syllabus

Introduction

Basics of CFD. Basics of mathematical background in CFD. Introduction to OpenFOAM, ParaView, LaTeX

CFD for Automotive Flows

Flow over a 2D car model. Basics of Linux. OpenFOAM on the terminal. Running simulations. Visualizing results.

CFD for Environmental Flows

Flow over a 3D urban area. Local parallel computing by OpenFOAM. Visualization on ParaView.

CFD for Thermo-fluid Flows

Cooling process of an electronic card. Theory begind the thermo-fluid analysis. Design optimization using OpenFOAM.

CFD for HVAC Systems

HVAC in a building. snappyHexMesh in OpenFOAM. Courant number and mesh convergence study.

CFD for Biological Flows

Simulation of heart pumping blood. Moving mesh method in OpenFOAM. Advanced visualization in ParaView. Non-newtonian fluids.

CFD for Multiphase Flows

Multiphase flow theory basis. Numerical methods for tracking interfaces between phases.

CFD for Fluid-structure Interaction

Coupling methods for FSI solvers. Response of structures to fluid flow. Structural mechanics eq.

CFD for Particle-laden Flows

Dynamics of particles in a fluid domain. Modeling and simulation of particulate suspension.

CFD for Turbomachineries

AMI and NRF methods in OpenFOAM. Basics of turbomachinery flow analysis. Writing Python scripts for post-processing.

CFD for Combustion

Fundamentals of combustion theory. Modeling chemical reactions in flows. Simulation of engines and furnaces.

CFD for Micro and Nano Flows

Fluid behavior at micro and nano scales. Slip boundary conditions and rarefaction efects.

CFD for Compressible Flows

High-speed aerodynamics. Shock waves and expansion fans. Supersonic and hypersonic flow simulations.

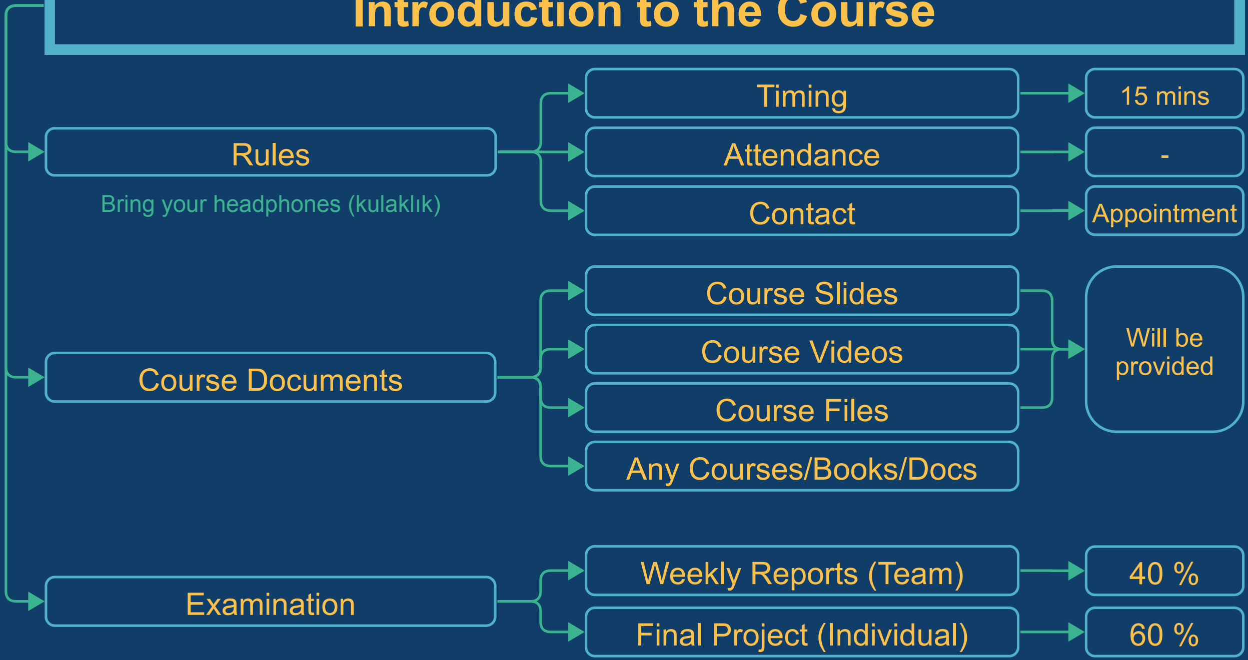
CFD for Aerospace Applications

Aerodynamic design and anlalysis of aircraft.

Introduction

- Introduction to the Course
- Introduction to CFD
- Introduction to OpenFOAM
- Introduction to ParaView
- Introduction to LaTeX

Introduction to the Course



- Rules
  - Timing → 15 mins
  - Attendance → -
  - Contact → Appointment

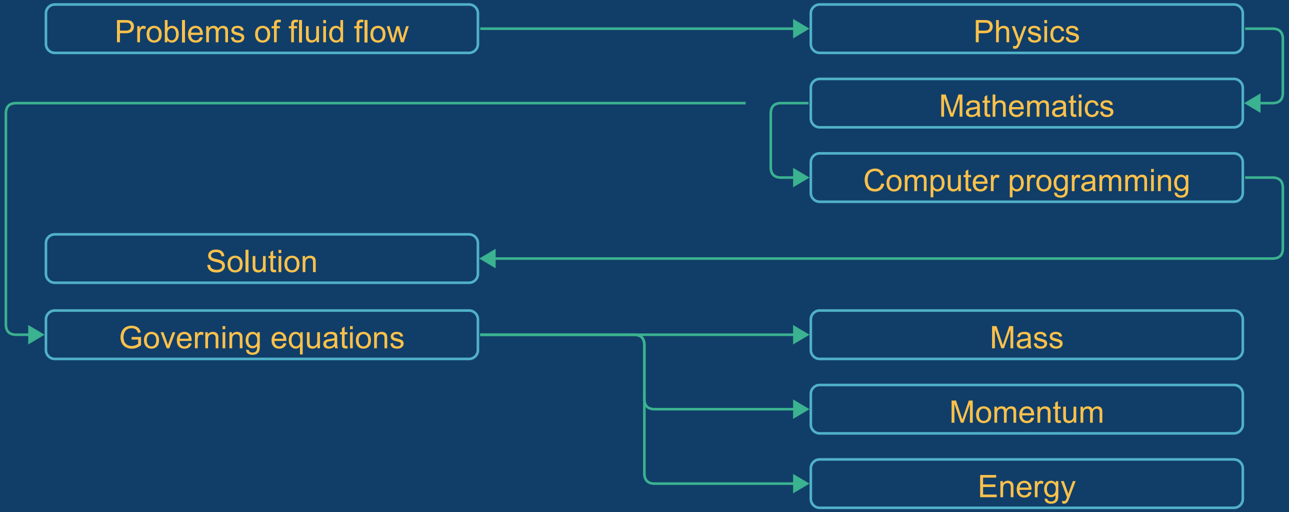
Bring your headphones (kulaklık)

- Course Documents
  - Course Slides
  - Course Videos
  - Course Files
  - Any Courses/Books/Docs

Course Slides, Course Videos, Course Files → Will be provided

- Examination
  - Weekly Reports (Team) → 40 %
  - Final Project (Individual) → 60 %

Introduction to CFD



Problems of fluid flow

Physics

Mathematics

Computer programming

Solution

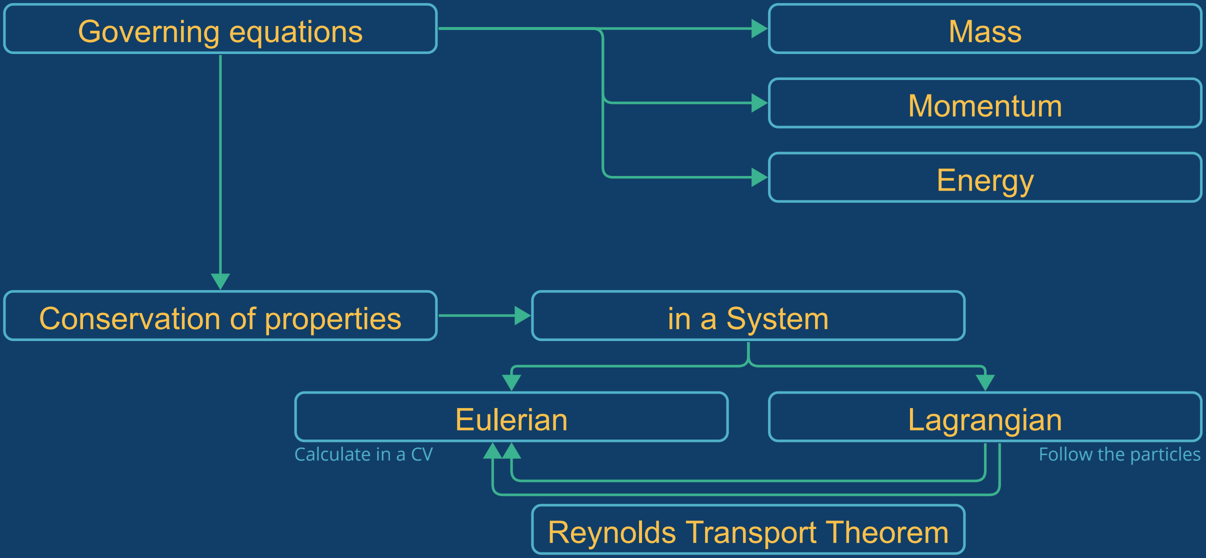
Governing equations

Mass

Momentum

Energy

Conservation of Quantities



Governing equations

Mass

Momentum

Energy

Conservation of properties

in a System

Eulerian

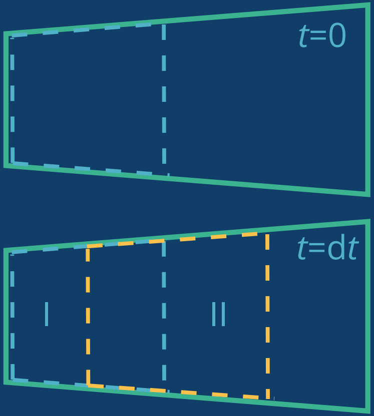
Calculate in a CV

Lagrangian

Follow the particles

Reynolds Transport Theorem

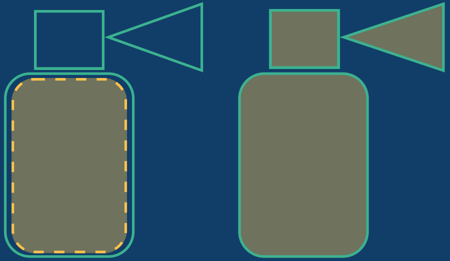
Eulerian and Lagrangian Methods



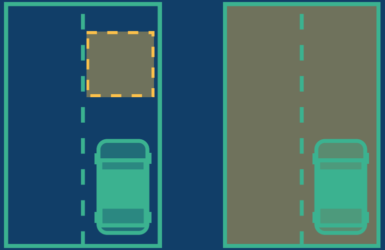
Control Volume
System

Control Volume
System

Bsis = Bcv + I - II



Spray



Moving car on a road

Reynolds Transport Theorem

$$\frac{\mathrm{d}B_{\mathrm{sys}}}{\mathrm{d}t} = \frac{\mathrm{d}}{\mathrm{d}t}\int_{\mathrm{cv}} \rho b dV + \oint_{\mathrm{cs}} \rho b(\mathbf{u}\cdot\mathbf{n})dA$$

$$b = \frac{B}{m}$$

B : Any extensive property
- mass
- momentum
- energy

b : Any intensive property
- density
- temperature
- velocity

sys : a system

cv : a control volume

cs : control surface of the cv

Governing Equations with RTT

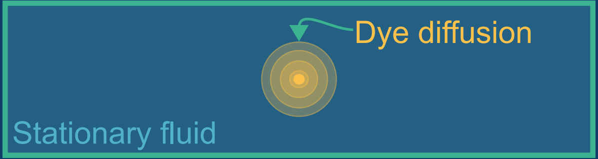
$$\frac{\mathrm{d}B_{\mathrm{sys}}}{\mathrm{d}t} = \frac{\mathrm{d}}{\mathrm{d}t}\int_{\mathrm{cv}} \rho b dV + \oint_{\mathrm{cs}} \rho b(\mathbf{u}\cdot\mathbf{n})dA$$

$$\text{mass} : \boldsymbol{\nabla}\cdot\mathbf{u} = 0$$

$$\text{momentum} : \rho\frac{\partial\mathbf{u}}{\partial t} + \rho\mathbf{u}\cdot\boldsymbol{\nabla}\mathbf{u} = -\boldsymbol{\nabla}p + \mu\boldsymbol{\nabla}^2\mathbf{u} + \rho\mathbf{g}$$

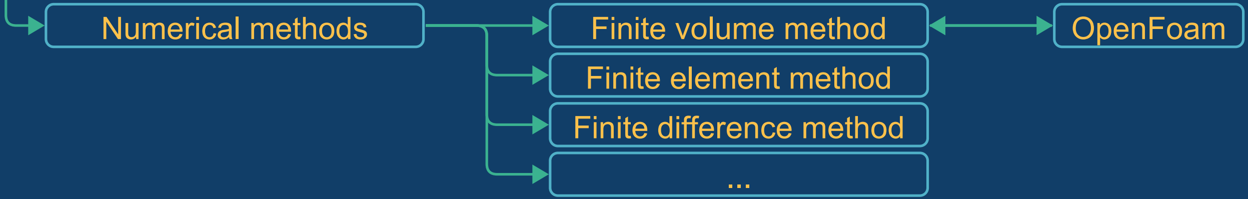
$$\text{energy} : \rho c_{\mathrm{p}}\frac{\partial T}{\partial t} + \rho c_{\mathrm{p}}\mathbf{u}\cdot\boldsymbol{\nabla}T = \kappa\boldsymbol{\nabla}^2 T + q$$

Transport Theorem



$$\begin{array}{rccccc}
\text{momentum}: & \rho\dfrac{\partial \mathbf{u}}{\partial t} & +\rho\mathbf{u}\cdot\nabla\mathbf{u} & = & \nabla\cdot\boldsymbol{\sigma} & +\rho\mathbf{g} \\
\text{energy}: & \rho c_{\mathrm{p}}\dfrac{\partial T}{\partial t} & +\rho c_{\mathrm{p}}\mathbf{u}\cdot\nabla T & = & \kappa\nabla^2 T & +q \\
\text{ADE}: & \rho\dfrac{\partial \phi}{\partial t} & +\rho\mathbf{u}\cdot\nabla\phi & = & \Gamma\nabla^2\phi & +\mathrm{S} \\
 & \underbrace{\qquad\qquad}_{\text{Temporal}} & \underbrace{\qquad\qquad\qquad}_{\text{Advection}} & & \underbrace{\qquad\qquad}_{\text{Diffusion}} & \underbrace{\qquad\quad}_{\text{Source}}
\end{array}$$

Solutions of the FM Governing Equations



Numerical methods

Finite volume method

Finite element method

Finite difference method

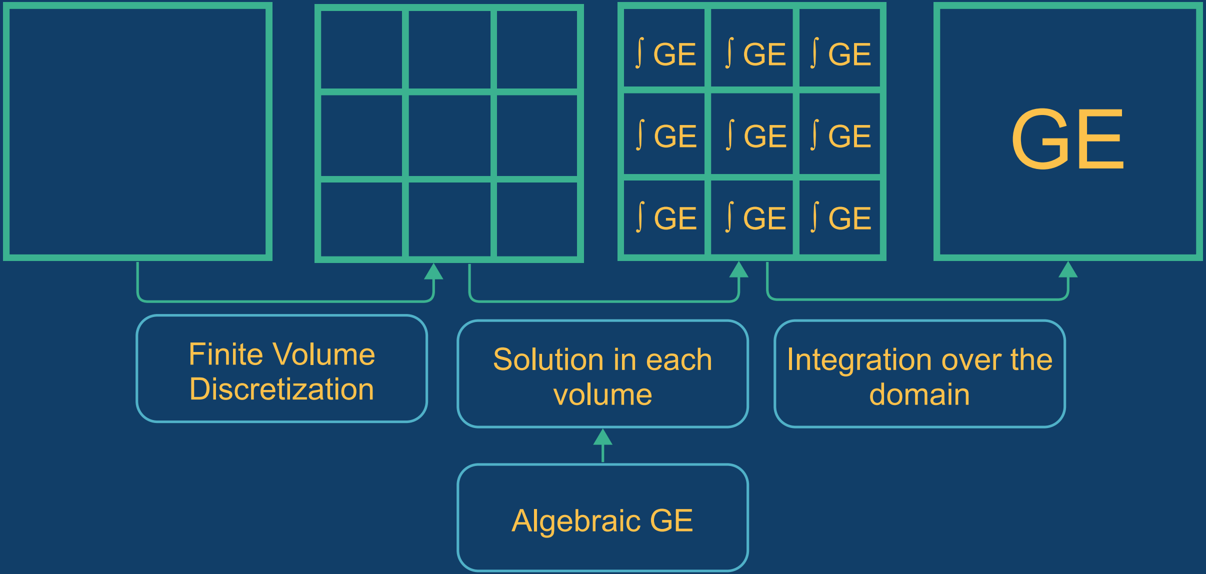
...

OpenFoam

New table

Feature	FVM	FVM	FDM
Geometry	Handles complex shapes	Adapts to complex shapes	Limited to simple shapes
Accuracy	High with refined mesh	Moderate with fine mesh	Lower compared to FEM/FVM
Implementation	More complex	Moderately complex	Simple
Computational Cost	Higher for large systems	Moderate	Lower
Suitability	Various PDEs, complex flows	Fluid dynamics, complex flows	Regular geometries, simple flows

Finite Volume Method



Finite Volume Discretization

Solution in each volume

Integration over the domain

Algebraic GE

GE: Governing equations

Finite Volume Method - 1D Example

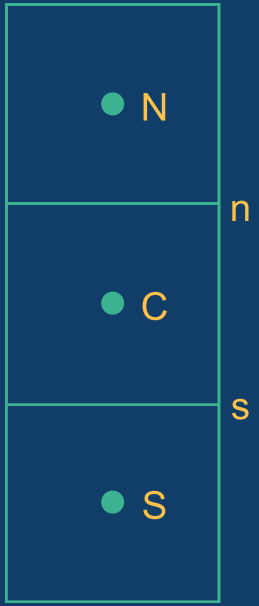
Conduction heat transfer problem

$$\kappa \nabla^2 T + q = 0$$

$$\frac{\mathrm{d}}{\mathrm{d}x}\left(\kappa \frac{\mathrm{d}T}{\mathrm{d}x}\right) + q = 0$$

$$\int_{\mathrm{s}}^{\mathrm{n}} \frac{\mathrm{d}}{\mathrm{d}x}\left(\kappa \frac{\mathrm{d}T}{\mathrm{d}x}\right) \mathrm{d}x + \int_{\mathrm{s}}^{\mathrm{n}} q \mathrm{d}x = 0$$

$$\boxed{\kappa_{\mathrm{s}}\left(\frac{T_{\mathrm{S}} - T_{\mathrm{N}}}{\Delta x_{\mathrm{S}}}\right) - \kappa_{\mathrm{n}}\left(\frac{T_{\mathrm{N}} - T_{\mathrm{S}}}{\Delta x_{\mathrm{N}}}\right) + q\Delta x = 0}$$



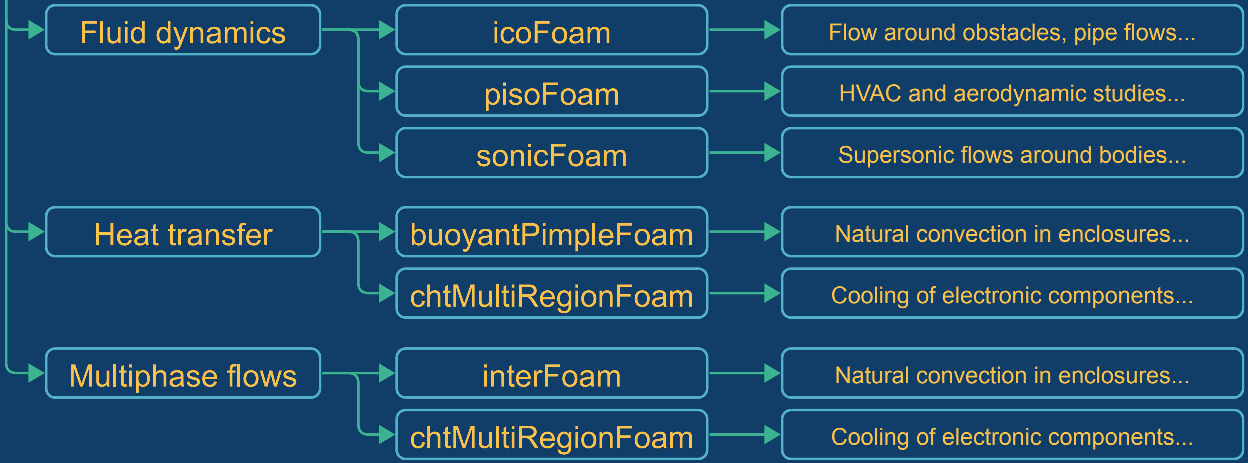
Introduction to OpenFOAM (1/2)

- OpenFOAM
  - **O**pen **F**ield **O**peration **A**nd **M**anipulation
- Features
  - Robust solver library
  - Pre/post-processing utilities
  - Customization and extensibility
- Applications
  - Fluid dynamics simulations
  - Heat transfer simulations
  - Chemical reactions, combustion and fire
  - Multiphase flows
  - Compressible flows
  - Fluid-structure interaction

Introduction to OpenFOAM (2/2)

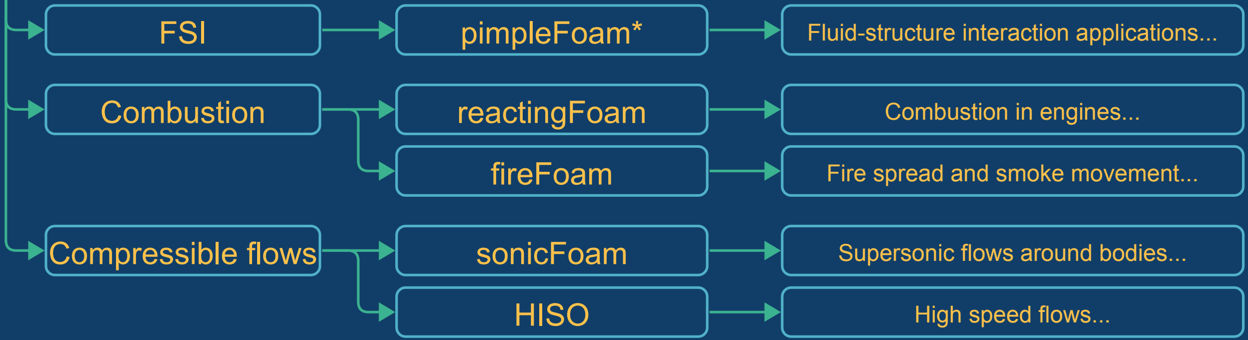
- Advandages
  - No licensing fees
  - Active community
  - Flexible and powerful
  - Extensive libraries
  - Continuous development
  - **Open source**
- Disadvantages
  - Limited GUI
  - Steep learning curve
  - Requires programming knowledge
  - Initial setup complexity

OpenFOAM Solvers (1/2)



- Fluid dynamics
  - icoFoam → Flow around obstacles, pipe flows...
  - pisoFoam → HVAC and aerodynamic studies...
  - sonicFoam → Supersonic flows around bodies...
- Heat transfer
  - buoyantPimpleFoam → Natural convection in enclosures...
  - chtMultiRegionFoam → Cooling of electronic components...
- Multiphase flows
  - interFoam → Natural convection in enclosures...
  - chtMultiRegionFoam → Cooling of electronic components...

OpenFOAM Solvers (2/2)



FSI

pimpleFoam*

Fluid-structure interaction applications...

Combustion

reactingFoam

Combustion in engines...

fireFoam

Fire spread and smoke movement...

Compressible flows

sonicFoam

Supersonic flows around bodies...

HISO

High speed flows...

Links

OpenFOAM
The OpenFOAM Foundation

CFD Direct
The Architects of OpenFOAM

ParaView

FreeCAD

The LaTeX Project

Gmsh

Links to software webpages

GitHub

YouTube

Links to course documentations and videos