

# OpenFOAM School – Module 01

## Introduction



EuroCC workshop

Aleksander GRM – May, 2024



# Outline

---

Short review

Small introduction into PDEs

Solution methods

Mesh description

Finite volume method

Conservation laws

Introduction to OpenFOAM

Structure

Basic details

Example

## Short review

---



- ▶ Small introduction into PDEs
- ▶ Solution methods
- ▶ Mesh description



The mathematical modelling of real systems is in most cases narrowed down to the mathematical model described by **Partial Differential Equations (PDE)**.

**Example:** PDE describing waves motion on a free surface

$$\boxed{\frac{\partial^2 u}{\partial t^2} = c^2 \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)} = c^2 \Delta u,$$

where the wave height is

$$u = u(t, \mathbf{x}), \quad \mathbf{x} \in \Omega, \quad \Omega \subseteq \mathbb{R}^2,$$

in the direction  $z$ , where  $z \perp \Omega$ .



In general we solve two types of problems

► **Initial Value problem (IVP):**

time  $t$  is independent variable of the problem, we solve **time-dependant** problem, so we need

*initial condition:*  $u_0 = u(t = t_0, \mathbf{x})$ , where  $\mathbf{x} \in \Omega \subseteq \mathbb{R}^n$

► **Boundary Value Problem (BVP):**

time  $t$  is not part of the problem, we solve **time-independent** problem, so we need

*boundary condition:*  $u_0 = u(\mathbf{x})$ , where  $\mathbf{x} \in \Gamma \subseteq \mathbb{R}^{n-1}$  ( $\Gamma = \partial\Omega$ ),

where  $\Gamma$  is **boundary** of the computational domain. Many times you may see  $\partial\Omega$ .

IVP condition is obtained with the solution of BVP (start BVP with intuitive initialization)!



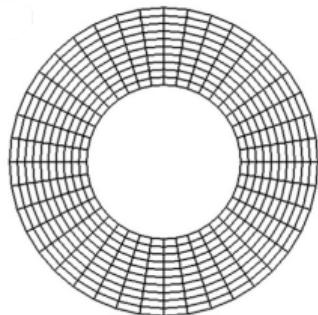
PDE can be solved in different ways

- ▶ **analytical methods:** mostly solving linear problems, or problems involving small parameter  $\epsilon$ . Methods used are: separation of variables, series expansion, Perturbation methods, Laplace transform, Complex analysis methods, ...
- ▶ **numerical methods:** solve problems that is not possible to solve with analytical methods. In general we distinguish:
  - ▶ finite difference method (FDM) - solving **strong** form
  - ▶ finite volume method (FVM) - solving **weak** form
  - ▶ finite element method (FEM) - solving **weak** form
- ▶ **special numerical approaches:** use of FDM, FVM and FEM in different combinations
  - ▶ Immersed Boundary Method - IBM
  - ▶ Smoothed Particle Hydrodynamics - SPH (mesh less method)



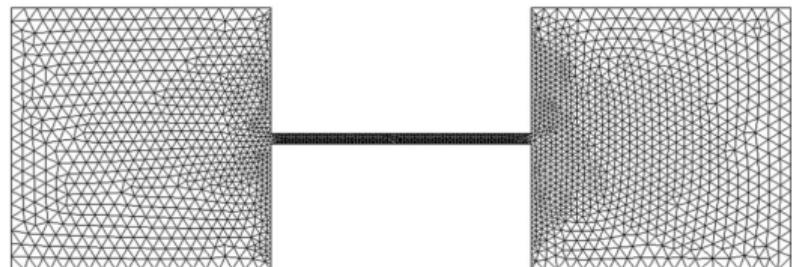
All numerical methods, except SPH, need the mesh. Mesh divides computational domain onto cells/elements

**Structured** grid



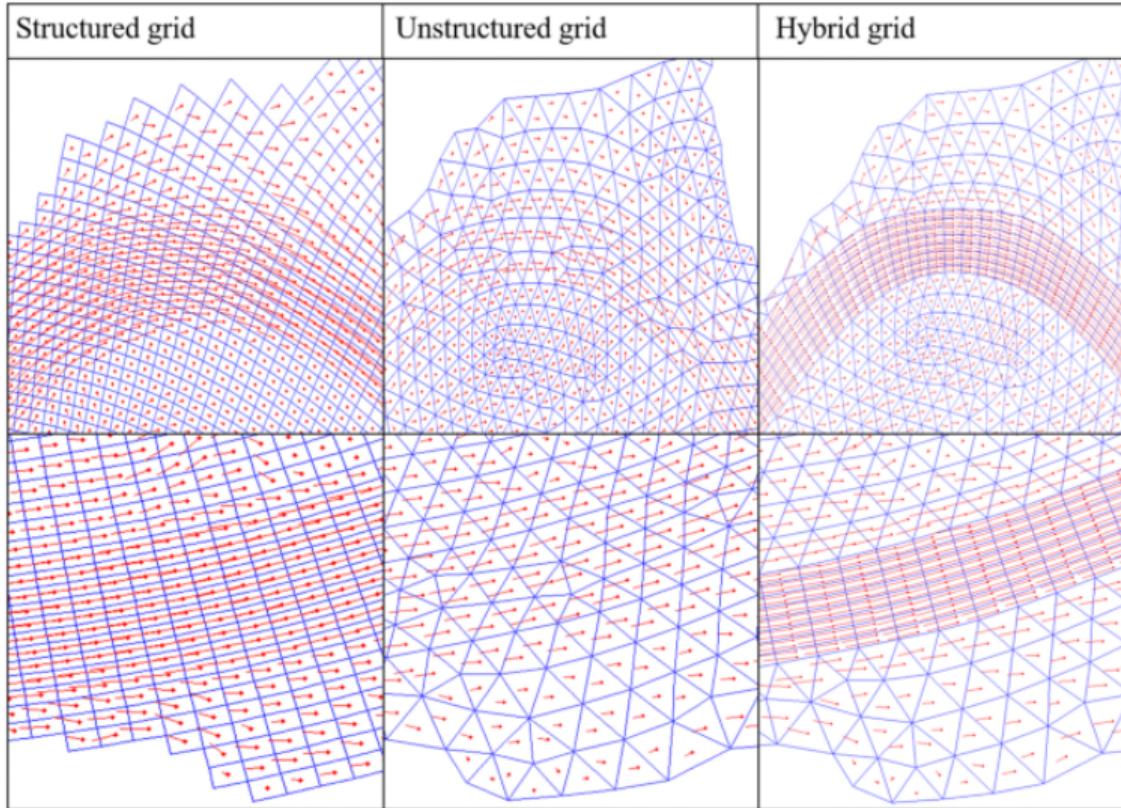
better convergence

**Unstructured** grid



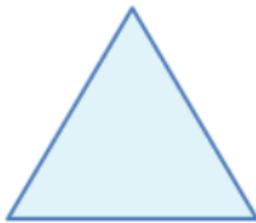
worse convergence

*Very often we have a combination of both types! (next pages)*





## 2D mesh

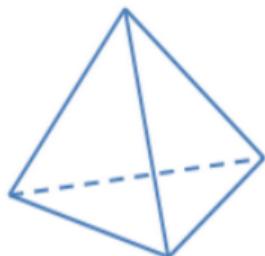


Triangle

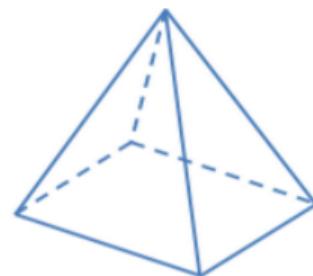


Quadrilateral

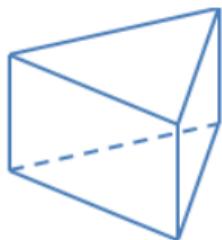
## 3D mesh



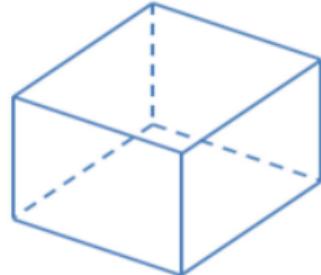
Tetrahedron



Pyramid

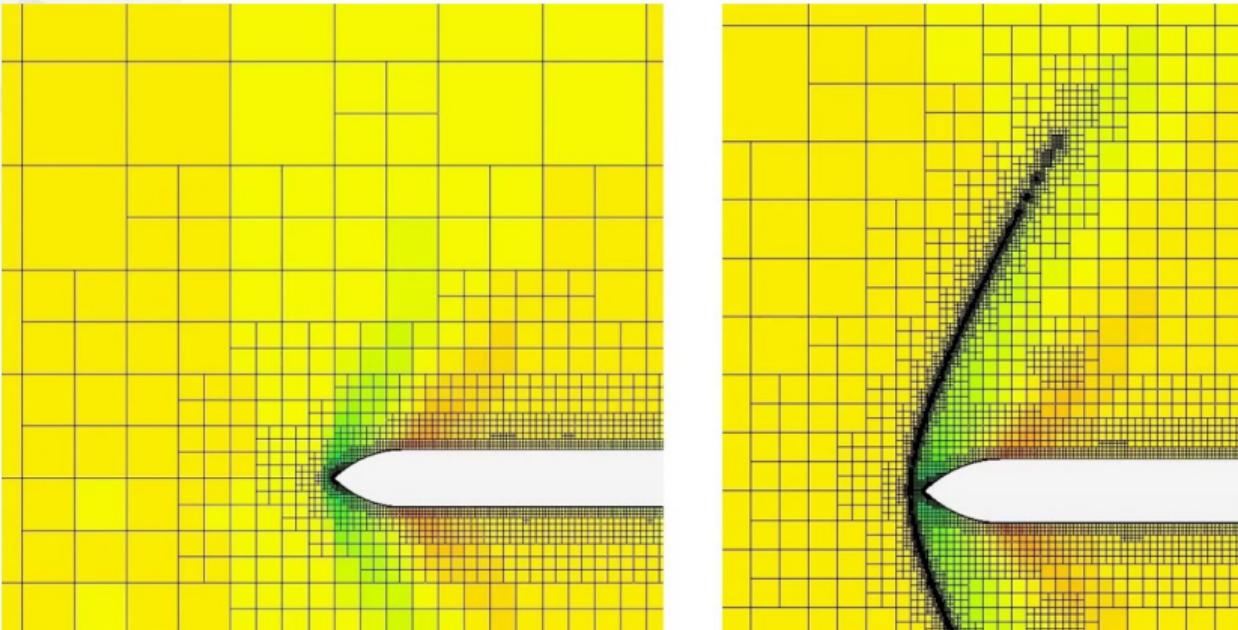


Triangular Prism

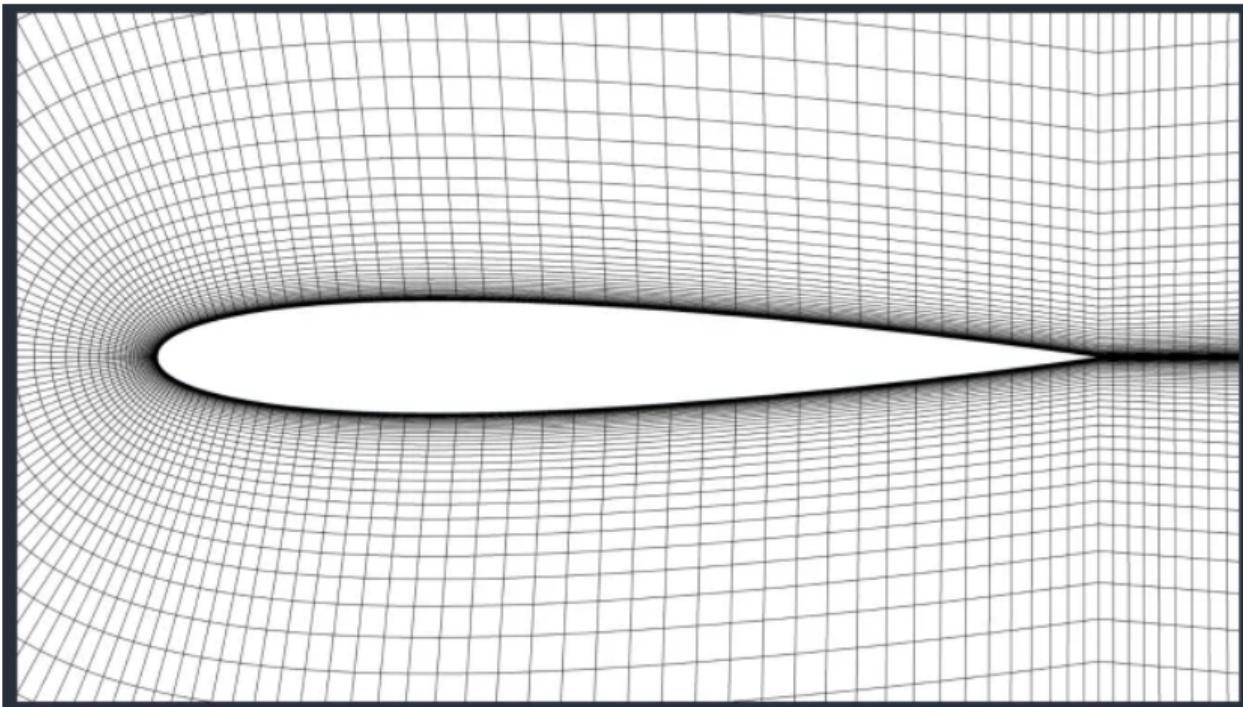


Hexahedron

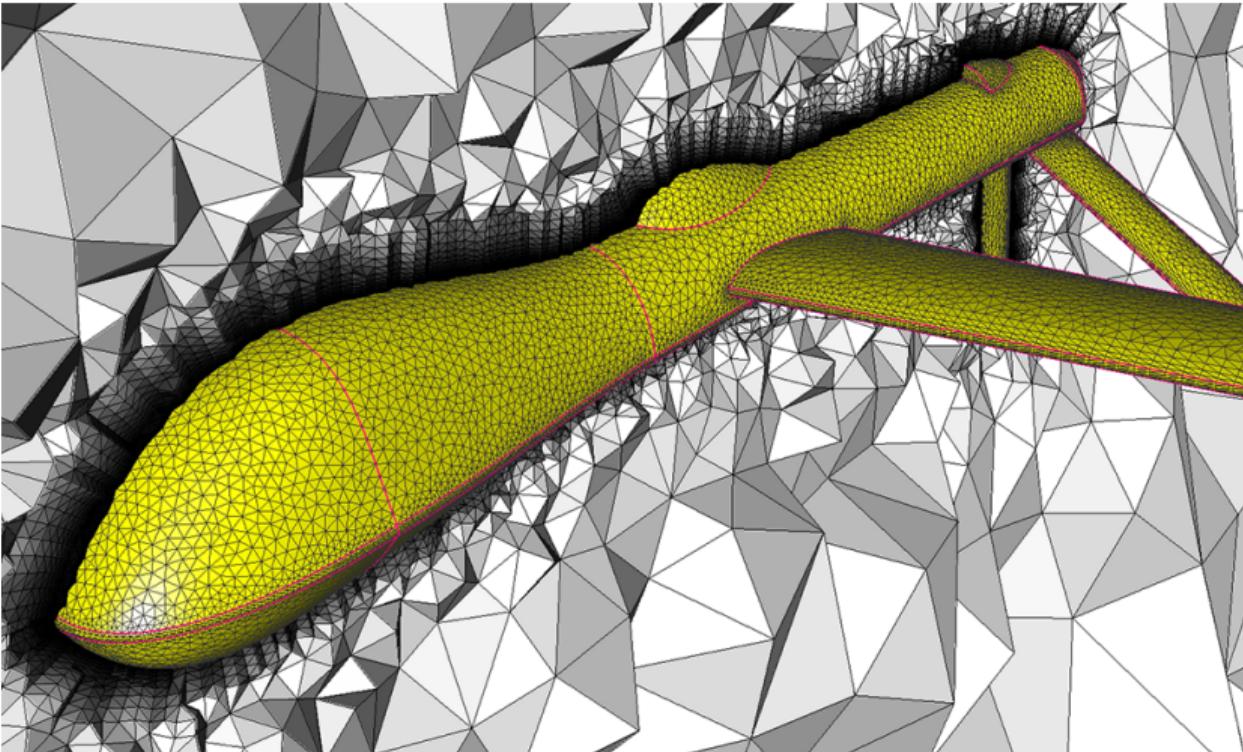
Some numerical solutions are only possible with **mesh refinement**!



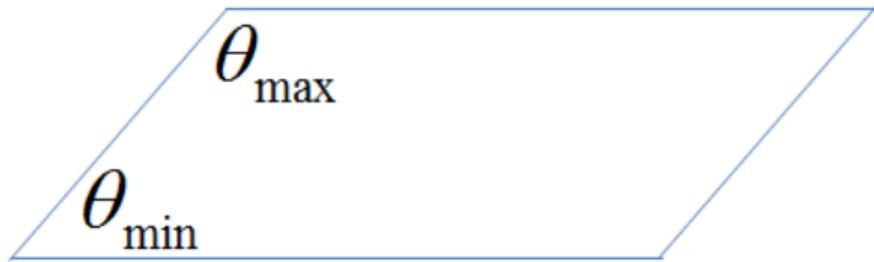
formation of shock waves - space entry simulation



external flow – foil geometry



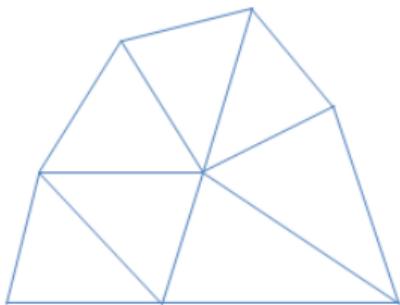
external flow – raptor geometry



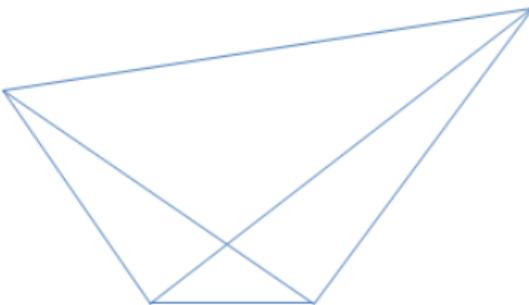
**Skewness** can be determined in many different ways, but always determines ratio of inner cell sides/angles!

$$\text{skewness} = \frac{\text{optimal cell size} - \text{cell size}}{\text{optimal cell size}}$$

It measures the deviation from optimal geometry (equidistant triangle)!



Smooth Change in cell size



Large jump in cell size

**Smoothness** measures the speed of cell size transition.



Aspect ratio = 1



High aspect ratio triangle



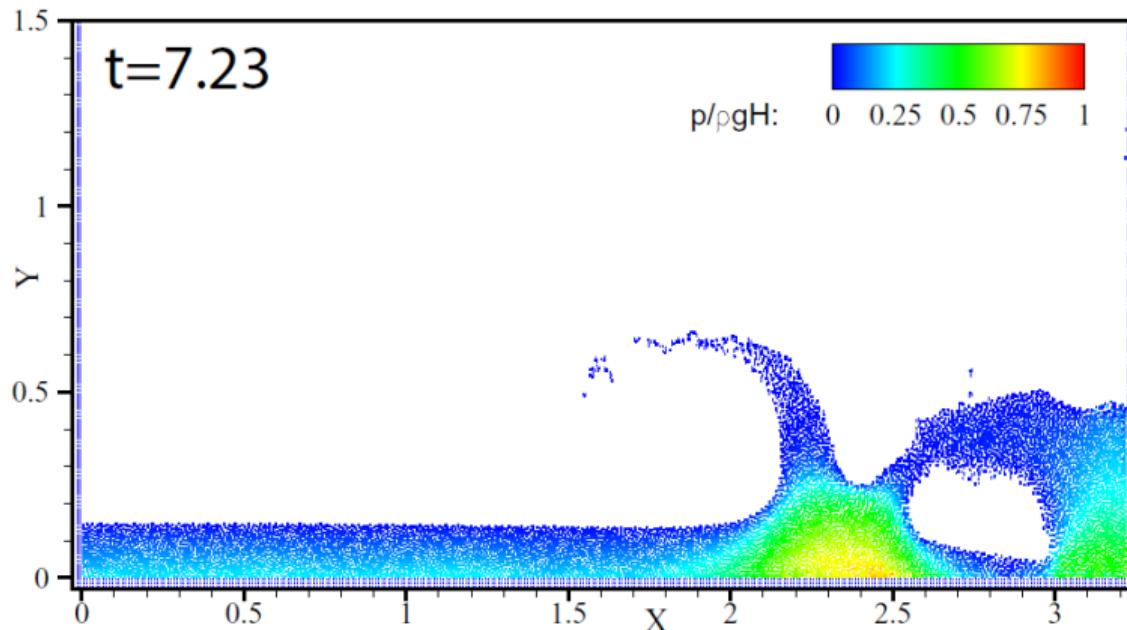
Aspect ratio = 1



High aspect ratio quad

**Aspect ratio** measures the ratio of longest to the shortest side in a cell. Ideally it should be equal to 1 to ensure best results.

In mesh-less methods, we use particles that fill the space. They're mainly used where the shape of the surface changes over time, e.g. waves, continuous casting, solidification, etc.



## Finite volume method

---



There are two types of reference frame describing the flow field

**Lagrange** frame of reference (moving frame with flow – **material volume** - MV)

The Lagrangian specification of the flow field is a way of looking at fluid motion where the observer follows an individual fluid parcel as it moves through space and time.

This can be visualized as sitting in a boat and drifting down a river.

**Euler** frame of reference (fixed frame – **control volume** - CV)

The Eulerian specification of the flow field is a way of looking at fluid motion that focuses on specific locations in the space through which the fluid flows as time passes.

This can be visualized by sitting on the bank of a river and watching the water pass the fixed location.

**Motion** in material volume (MV) is described by the mapping  $\mathbf{x}(t, \mathbf{x}_0)$ , that maps particle  $\mathbf{x}_0$  from initial position in time  $t_0$  to future position in time  $t$

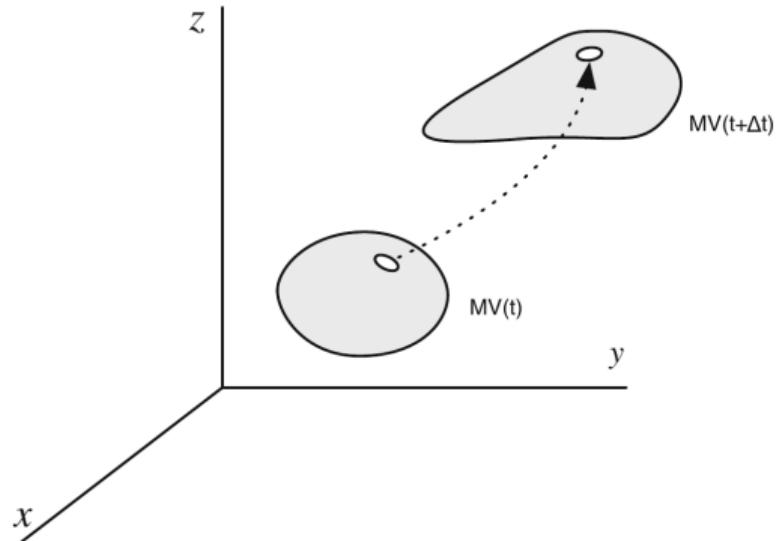
$$\boxed{\mathbf{x}(t, \mathbf{x}_0) : (t_0, \mathbf{x}_0) \rightarrow (t, \mathbf{x}_0)}$$

describing the particle  $\mathbf{x}_0$  path in time  $t$  (**path line**).

---

Flow velocity can be found as

$$\boxed{\mathbf{v}(t, \mathbf{x}(t, \mathbf{x}_0)) = \frac{\partial}{\partial t} \mathbf{x}(t, \mathbf{x}_0)}$$



Relation between the motion in Lagrange and Euler reference frame.



The derivative (rate of change) of a field variable  $\phi(t, \mathbf{x}(t))$

with respect to fixed position

$$\frac{\partial}{\partial t} \phi(t, \mathbf{x}(t))$$

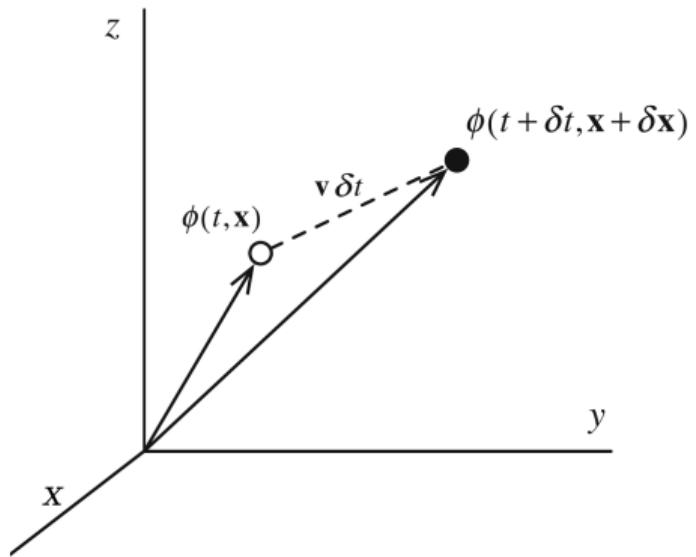
is called Euler derivative.

---

the derivative following a moving fluid parcel

$$\frac{D}{Dt} \phi(t, \mathbf{x}(t)) = \left( \frac{\partial}{\partial t} + (\mathbf{v} \cdot \nabla) \right) \phi$$

is called Lagrange derivative.





$$\begin{aligned}\text{Term I} &= \frac{d}{dt} \left[ \int_{\Omega(t)} (\rho \phi) dV \right] \\ &= \int_{\Omega} \left[ \frac{\partial}{\partial t} (\rho \phi) + \nabla \cdot (\rho \mathbf{v} \phi) \right] dV = \int_{\Omega} \left[ \frac{D}{Dt} (\rho \phi) + \rho \phi \nabla \cdot \mathbf{v} \right] dV\end{aligned}$$

where  $\Omega(t)$  is **material volume** and  $\Omega$  is **control volume** and  $\mathbf{v}$  is fluid velocity.

Change of the  $\phi$   
over time  $\Delta t$  within  
the material volume

**Term I**

=

Surface flux of the  $\phi$   
over time  $\Delta t$  across  
the control volume

**Term II**

+

Source/Sink of  $\phi$   
over time  $\Delta t$  within  
the control volume

**Term III**



The most general PDE (scalar or vector) encountered in fluid flow is

$$\frac{\partial \phi}{\partial t} + \nabla \cdot \mathbf{F}(\phi) = Q(\phi)$$

- ▶ **fluxes:**  $\mathbf{F}(\phi)$
- ▶ **sources:**  $Q(\phi)$

showing **conservation law**. It will be used to demonstrate the FVM discretization process!

**Fluxes** are normally two

- ▶ **advection:**  $\mathbf{F}_A(\phi) = \mathbf{v}\phi$
- ▶ **diffusion:**  $\mathbf{F}_D(\phi) = -D(\phi) \nabla \phi$



Change of the  $\phi$  over time  $\Delta t$  within the material volume

=

Surface flux of the  $\phi$  over time  $\Delta t$  across the control volume

+

Source/Sink of  $\phi$  over time  $\Delta t$  within the control volume

**Term I**

**Term II**

**Term III**

$$\int_{\Omega} \left[ \frac{\partial \phi}{\partial t} \right] dV$$

=

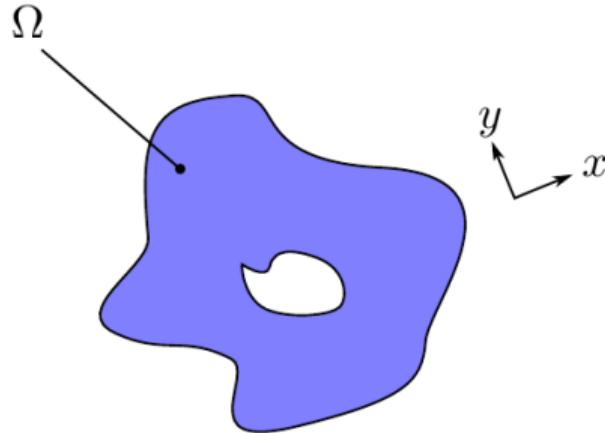
$$\int_{\Omega} [\nabla \cdot \mathbf{F}(\phi)] dV$$

+

$$\int_{\Omega} Q(\phi) dV$$

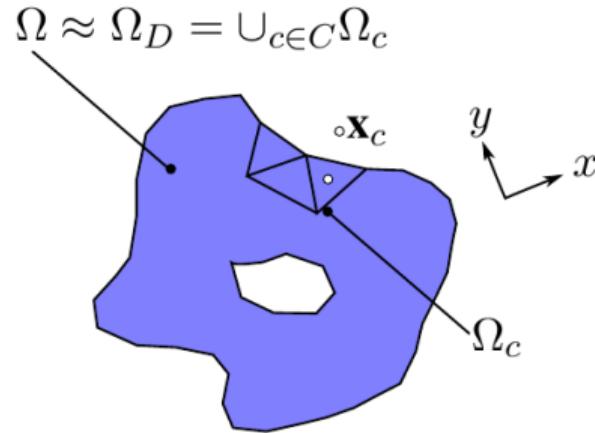
The conservation equation for ADS PDE in fluid flow can be expressed as

$$\int_{\Omega} \left[ \frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \mathbf{v}) \right] dV = \int_{\Omega} \nabla \cdot (D(\phi) \nabla \phi) dV + \int_{\Omega} Q(\phi) dV$$



$$\phi = \phi(\mathbf{x}, t)$$

(a) Continuous flow domain.



$$\phi(\mathbf{x}, t) \doteq \phi(\mathbf{x}_c) + \nabla \phi(\mathbf{x}_c)(\mathbf{x} - \mathbf{x}_c)$$

(b) Discretized flow domain.

$$\phi(\mathbf{x}) = \phi(\mathbf{x}_c) + \nabla \phi(\mathbf{x}_c)(\mathbf{x} - \mathbf{x}_c) + \nabla \nabla \phi(\mathbf{x}_c) : (\mathbf{x} - \mathbf{x}_c) \otimes (\mathbf{x} - \mathbf{x}_c) + \mathcal{O}(\|\mathbf{x} - \mathbf{x}_c\|^3)$$



Expressing volume average over  $\Omega_c$  introduces averaged variable  $\phi_c$  over  $\Omega_c$  at the point  $\mathbf{x}_c$ , which are constant over cell  $\Omega_c$

$$\begin{aligned}\phi_c = & \phi(\mathbf{x}_c) \\ & + \frac{1}{|\Omega_c|} \nabla \phi(\mathbf{x}_c) \int_{\Omega_c} (\mathbf{x} - \mathbf{x}_c) \, dV \\ & + \frac{1}{|\Omega_c|} \nabla \nabla \phi(\mathbf{x}_c) : \int_{\Omega_c} (\mathbf{x} - \mathbf{x}_c) \otimes (\mathbf{x} - \mathbf{x}_c) \, dV + \dots\end{aligned}$$

by the definition of  $\mathbf{x}_c$  to be the **cell centre** it follows

$$\int_{\Omega_c} (\mathbf{x} - \mathbf{x}_c) \, dV = 0.$$

The **average value** of  $\phi$  over the finite volume  $\Omega_c$  is exactly equal to the value of  $\phi$  at the centroid  $\mathbf{x}_c$  of  $\Omega_c$  for a **linear**  $\phi$  (method is at least 2<sup>nd</sup> order).



$$\phi_c \approx \frac{1}{|\Omega_c|} \int_{\Omega_c} \phi_{\text{linear}}(\mathbf{x}_c) dV + \mathcal{O}(\|\mathbf{x} - \mathbf{x}_c\|^2)$$

Cell-centered FVM is at least 2<sup>nd</sup> order method

The domain discretization of the unstructured FVM that assigns cell-average values  $\phi_c$  of  $\phi$  at centroids  $\mathbf{x}_c$  of the cells  $\Omega_c$  is **second-order accurate**.

The average value of  $\phi$  over the finite volume  $\Omega_c$  is exactly equal to the value of  $\phi$  at the centroid  $\mathbf{x}_c$  of  $\Omega_c$  for a linear  $\phi$ , because for a linear  $\phi$  the higher-order derivatives are zero. In other words, the cell-average (cell-centered) value at the centroid of finite volumes recovers values of linear fields exactly. A method that exactly recovers values of linear functions is at least second-order accurate.



$$\int_{\Omega_c} \frac{\partial \phi}{\partial t} dV = \left( \frac{\partial \phi}{\partial t} \right)_c |\Omega_c| + \mathcal{O}(\|\mathbf{x} - \mathbf{x}_c\|^2)$$

we use **finite difference** to approximate temporal term

$$\text{backward Euler: } \left( \frac{\partial \phi}{\partial t} \right)_c^{n+1} = \frac{\phi_c^{n+1} - \phi_c^n}{\delta t} + \mathcal{O}(\delta t)$$

$$\text{BDS2: } \left( \frac{\partial \phi}{\partial t} \right)_c^{n+1} = \frac{3\phi_c^{n+1} - 4\phi_c^n + \phi_c^{n-1}}{2\delta t} + \mathcal{O}(\delta t^2)$$

where  $n+1$  is new time step,  $n$  is current time step and  $n-1$  is previous time step.

Temporal term is discretized with special FD scheme (e.g. BDS2)!



Convert volume integral to surface integral, using divergence theorem

$$\int_{\Omega_c} \nabla \cdot (\phi \mathbf{v}) dV = \int_{\partial\Omega_c} \phi \mathbf{v} \cdot \mathbf{n} dS.$$

In the case if many surfaces enclose the volume  $\Omega_c$  and forms surface enclosure  $\partial\Omega_c$  we can write

$$\partial\Omega_c = \bigcup_{f \in F_s} S_f,$$

where  $F_c$  is the index set of the faces  $S_f$  of the cell  $\Omega_c$ . The integral can be transformed into a sum over all cell surfaces  $S_f$

$$\int_{\partial\Omega_c} \phi \mathbf{v} \cdot \mathbf{n} dS = \sum_{f \in F_c} \int_{S_f} \phi \mathbf{v} \cdot \mathbf{n} dS.$$



$$\int_{\partial\Omega_c} \phi \mathbf{v} \cdot \mathbf{n} dS = \sum_{f \in F_c} \int_{S_f} \phi \mathbf{v} \cdot \mathbf{n} dS.$$

Averaging  $\phi$  over surface  $S_f$  in face centre  $\mathbf{x}_f$  we obtain

$$\phi_f = \frac{1}{|S_f|} \int_{S_f} \phi dS + \mathcal{O}(\|\mathbf{x} - \mathbf{x}_c\|^2),$$

where  $\phi_f = \phi(\mathbf{x}_f)$ . The advection term discretization reduces to

$$\boxed{\int_{\partial\Omega_c} \phi \mathbf{v} \cdot \mathbf{n} dS = \sum_{f \in F_c} \phi_f \mathbf{v}_f \cdot \mathbf{s}_f + \mathcal{O}(\|\mathbf{x} - \mathbf{x}_c\|^2)}$$

The rest of terms is discretized in a similar way!

(Look in the free book **The OpenFOAM – Technology Primer**)

# Introduction to OpenFOAM

---



- ▶ OpenFOAM folder structure
- ▶ Literature and Basic details
- ▶ Introductory example



Directory structure of OpenFOAM system as downloaded from a **GIT** repository

```
OF system folder
├── applications -- source code for solvers
├── bin -- bash scripts
├── doc -- documentation
├── etc -- compile & runtime controls
├── platforms -- platform specific compiled binaries
├── src -- source codes of the system
├── tutorials -- pre-configured cases
└── wmake -- compile script system
```



Files containing initial condition for all dependant variables

- 0
  - U -- velocity
  - p -- pressure
  - k -- turbulent
  - epsilon -- turbulent
  - T -- scalar transport



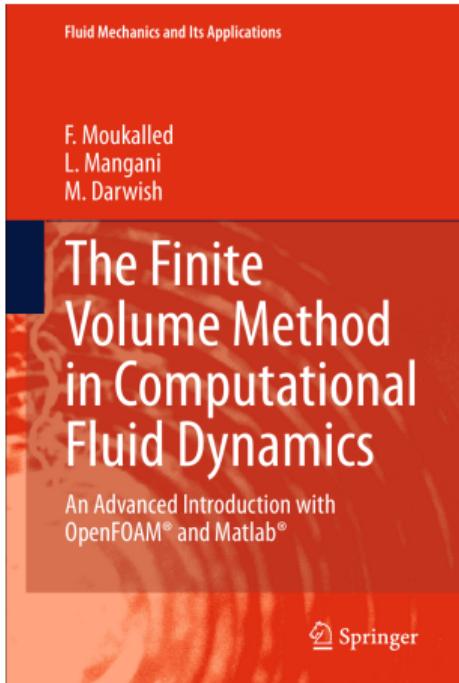
Folder containing files with constant data and mesh

```
constant
└── turbulenceProperties -- turbulent model properties
└── physicalProperties -- viscosity model & flow type
└── polyMesh -- computational mesh
    ├── boundary
    ├── points
    ├── faces
    ├── owner
    └── neighbour
```

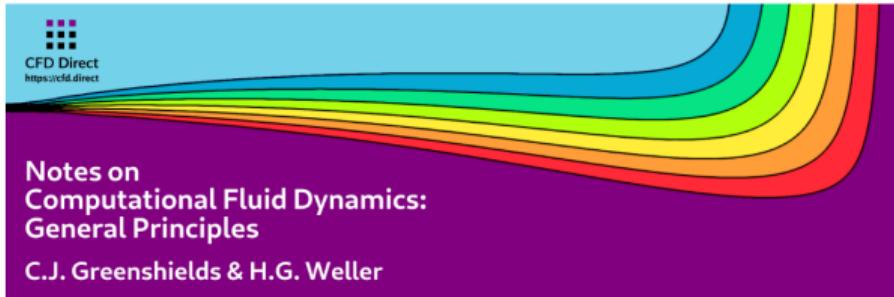


Folder containing system files

```
system
└── controlDict -- simulation controls
└── fvSchemes -- discretization schemes
└── fvSolution -- solution procedures
└── decomposeParDict -- domain decomposition & parallelization
└── residuals -- simulation residuals for post-process
└── ...
```



[link to the book](#)



#### About the Book

*Notes on Computational Fluid Dynamics* (CFD) was written for people who use CFD in their work, research or study, providing essential *knowledge* to perform CFD analysis with confidence. It offers a modern perspective on CFD with the finite volume method, as implemented in OpenFOAM and other popular general-purpose CFD software. Fluid dynamics, turbulence modelling and boundary conditions are presented alongside the numerical methods and algorithms in a series of short, digestible topics, or *notes*, that contain complete, concise and relevant information. The book benefits from the experience of the authors: Henry Weller, core developer of OpenFOAM since writing its first lines in 1989; and, Chris Greenshields, who has delivered over 650 days of CFD training with OpenFOAM.

#### Contents

[Preface](#)

[Symbols](#)

[1 Introduction](#)

[2 Fluid Dynamics](#)

[3 Numerical Method](#)

[4 Boundary Conditions](#)

[5 Algorithms and Solvers](#)

[6 Introduction to Turbulence](#)

[7 Reynolds-Averaged Turbulence Modelling](#)

[8 Sample Problems](#)

[Index](#)

ISBN 978-1-3999-2078-0, 291 pages.

[link to the book](#)



Dimensions in OF are set in a list

No.	Property	SI unit	USCS unit
1	Mass	kilogram (kg)	pound-mass (lbm)
2	Length	metre (m)	foot (ft)
3	Time	second (s)	second (s)
4	Temperature	Kelvin (K)	degree Rankine ( $^{\circ}$ R)
5	Quantity	mole (mol)	mole (mol)
6	Current	ampere (A)	ampere (A)
7	Luminous intensity	candela (cd)	candela (cd)

Example: kinematic viscosity  $\nu$  [ $\text{m}^2/\text{s}$ ]

- value is set in a file `constants/transportProperties`

```
nu [0 2 -1 0 0 0 0] 0.01;
```



## OF diversity

- ▶ many turbulent flows ( $k-\varepsilon$ ,  $k-\omega$ ,  $k-\omega$ -SST,...)
- ▶ many boundary conditions
- ▶ multi phase flows models
- ▶ internal combustion models
- ▶ DNS
- ▶ stress analysis + FSI
- ▶ and many more ...
- ▶ look into [www.openfoam.org](http://www.openfoam.org) - User Guide



## Example of a simple case in OF – Cavity flow

1. create mesh: `blockMesh`

2. check mesh quality: `checkMesh`

3. run solver: `foamRun`

4. check residuals:

```
foamMonitor -l postPorocessing/residuals/0/residuals.dat
```

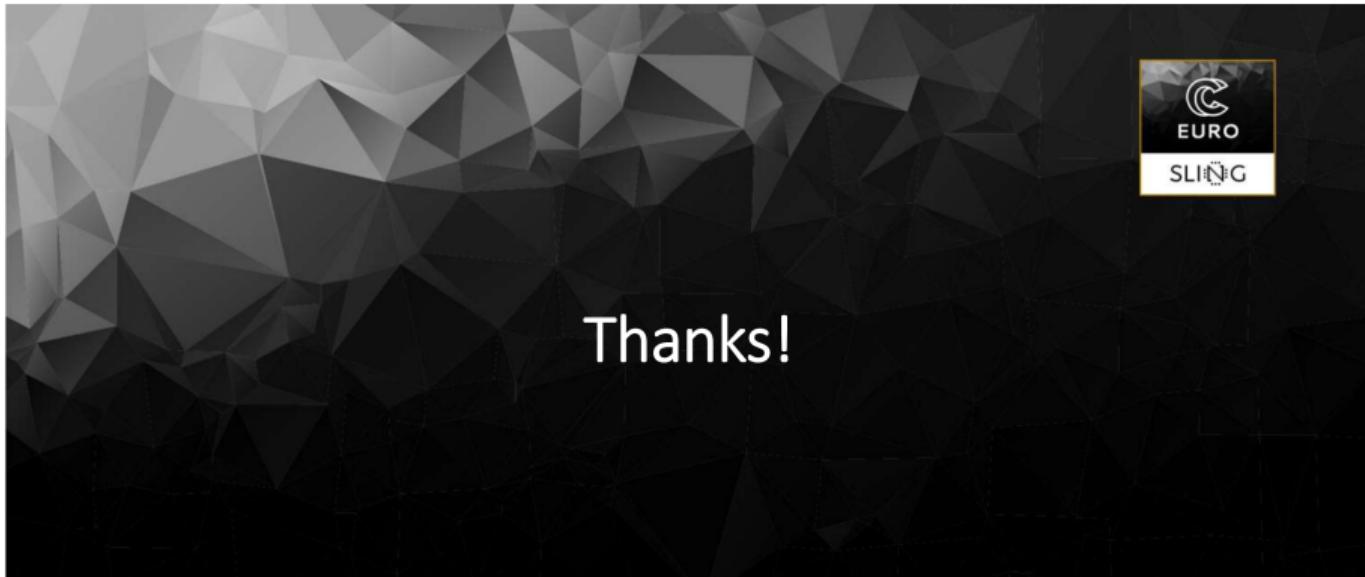
5. preview results: `paraFoam -builtin`

Show and try in **HPC@FS** system!



# Sponsorship

36/36



This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 951732. The JU receives support from the European Union's Horizon 2020 research and innovation programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, United Kingdom, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Switzerland, Turkey, Republic of North Macedonia, Iceland, Montenegro



**EuroHPC**  
Joint Undertaking