



Science and  
Technology  
Facilities Council

Scientific Computing



**CCP-WSI**  
a Collaborative Computational Project  
in Wave Structure Interaction

# OpenFOAM Fundamentals

Dr Omar Mahfoze

[omar-ahmed.mahfoze@stfc.ac.uk](mailto:omar-ahmed.mahfoze@stfc.ac.uk)

## OpenFOAM Parallel Performance Engineering Workshop

[Register](#) [Agenda](#)

5 - 6 June 2023

Time TBC

Daresbury Laboratory, Keckwick Lane, WA4 4AD

A photograph of a server rack with multiple units. The OpenFOAM logo is visible in the bottom left corner of the image.

OpenFOAM

# Table of Content

- **What is OpenFOAM?**
- **OpenFOAM Structure**
- **Numerics of OpenFOAM**
  - **Finite Volume Method**
  - **Time Discretisation**
  - **Pressure Velocity Coupling**
  - **Volume of Fluid**



Science and  
Technology  
Facilities Council

Scientific Computing

# What is OpenFOAM?



Science and  
Technology  
Facilities Council

Scientific Computing



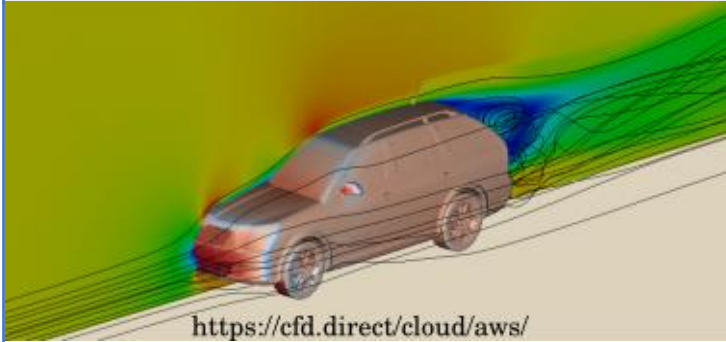
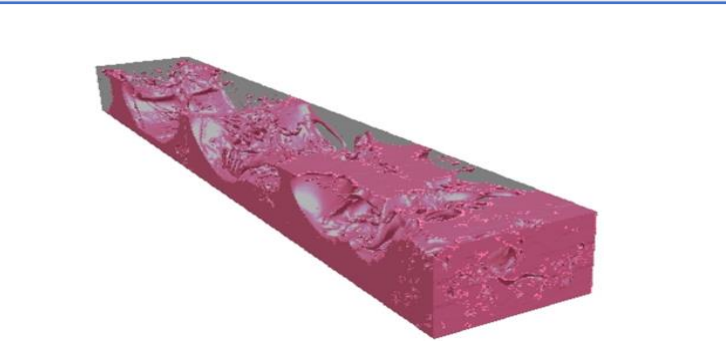
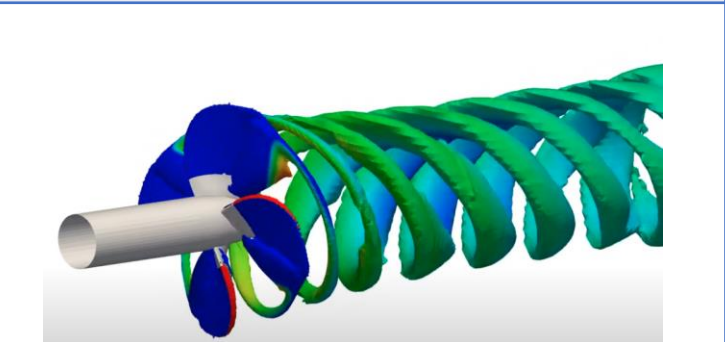
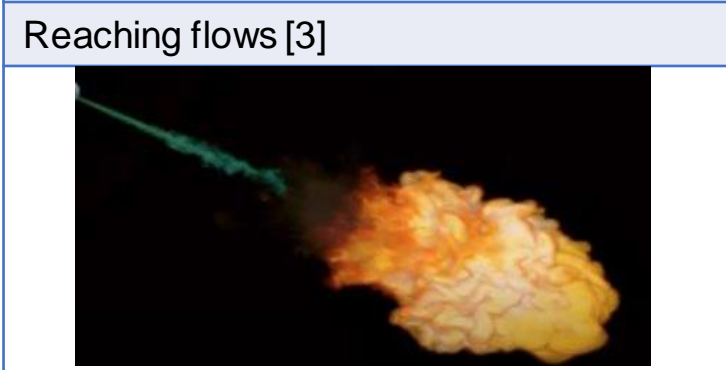
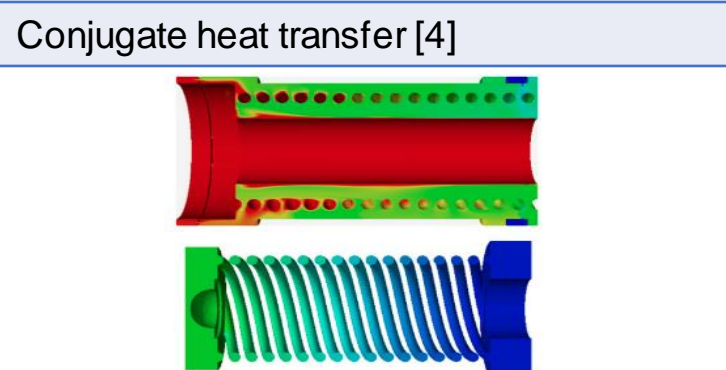
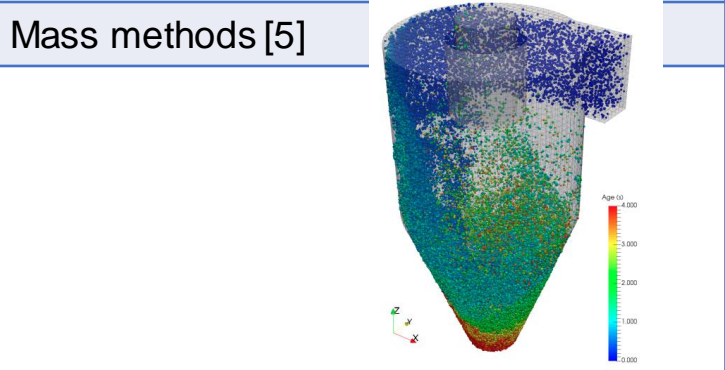
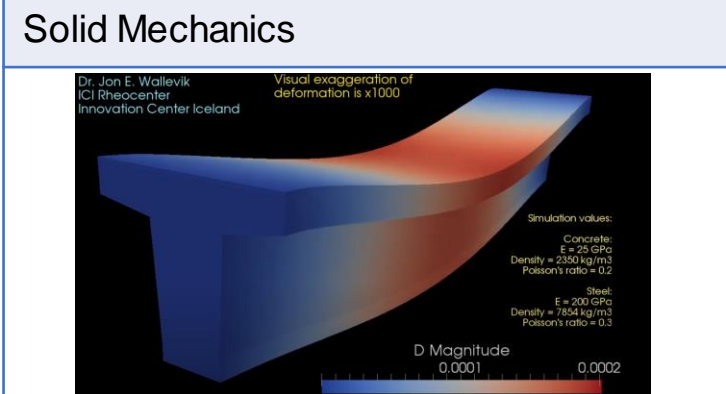
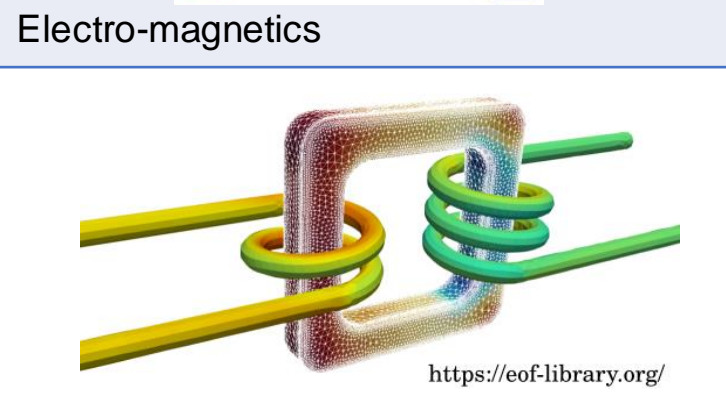
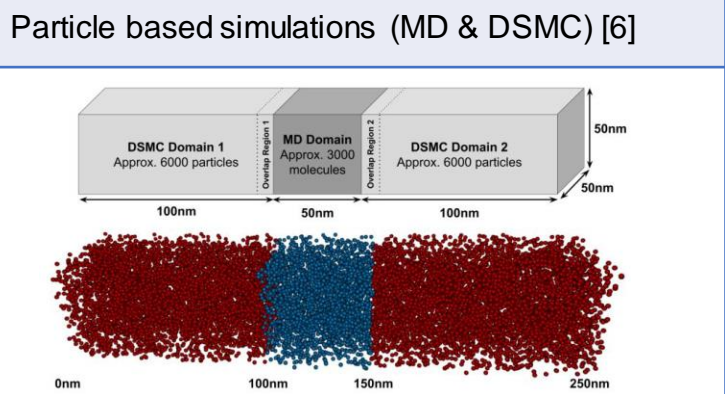
# What is OpenFOAM

- FOAM stands for "Field Operation and Manipulation"
- It is a library of tools written in C++ with high utilisation of Object Oriented Programming
- There are different versions:
  - OpenFOAM ESI (OpenFOAM-v20xx) ([www.openfoam.com](http://www.openfoam.com))
  - OpenFOAM foundation(OpenFOAM x) ([www.openfoam.org](http://www.openfoam.org))
  - OpenFOAM Extend (<https://github.com/Unofficial-Extend-Project-Mirror>)
- The core solvers are similar
- Switching between solvers is easy, giving the capabilities are available in both versions.
- There are some differences
  - ESI has more solvers and utilities (overset method, interIsoFoam)
  - ESI do more frequent releases
  - Foundation has a more generalized implementation
  - Some solvers gives different results

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



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

Single Phase flow	MultiPhase flow [1]	Turbo-machinery [2]
 <p><a href="https://cfd.direct/cloud/aws/">https://cfd.direct/cloud/aws/</a></p>		
Reaching flows [3]	Conjugate heat transfer [4]	Mass methods [5]
		
Solid Mechanics	Electro-magnetics	Particle based simulations (MD & DSMC) [6]
 <p>Dr. Jon E. Wallevik ICI Rissnescenter Innovation Center Iceland</p> <p>Visual exaggeration of deformation is x1000</p> <p>Simulation values: Concrete: E = 25 GPa Density = 2350 kg/m<sup>3</sup> Poisson's ratio = 0.2 Steel: E = 200 GPa Density = 7854 kg/m<sup>3</sup> Poisson's ratio = 0.3</p> <p>D Magnitude 0.0001 0.0002</p>	 <p><a href="https://eef-library.org/">https://eef-library.org/</a></p>	

[1] <https://www.mdpi.com/2076-3417/12/17/8481>

[3] <https://ecn.sandia.gov/diesel-spray-combustion/target-condition/spray-ab/>

[5] [https://www.foamacademy.com/wp-content/uploads/2016/11/GOFUN2017\\_ParticleSimulations\\_slides.pdf](https://www.foamacademy.com/wp-content/uploads/2016/11/GOFUN2017_ParticleSimulations_slides.pdf)

[6] <https://www.sciencedirect.com/science/article/pii/S0045793020302966>

[2] [https://www.youtube.com/watch?v=9eNYtOQlixY&ab\\_channel=EngineerDo](https://www.youtube.com/watch?v=9eNYtOQlixY&ab_channel=EngineerDo)

[4] <https://www.sciencedirect.com/science/article/pii/S1359431117349840>



Science and  
Technology  
Facilities Council

Scientific Computing

# OpenFOAM Structure



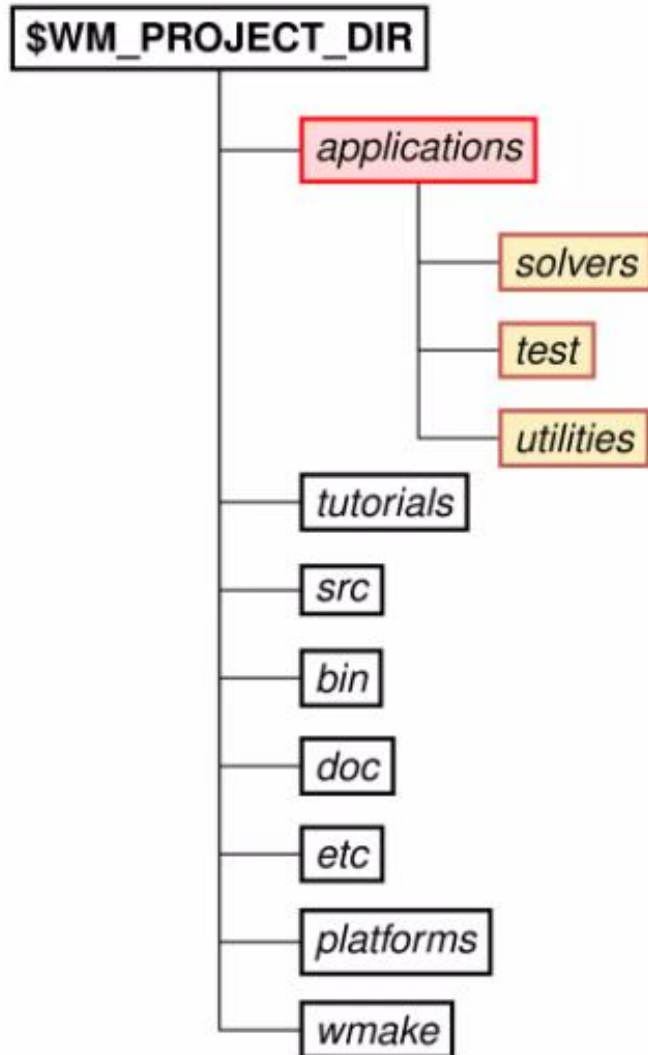
Science and  
Technology  
Facilities Council

Scientific Computing





# OpenFOAM Structure



## 1. *applications*:

alias: *app* = 'cd \$FOAM\_APP'.

This directory contains the source files of all the executables created using the **C++** libraries. It contains the following directories:

### 1.1 *solvers*:

alias: *sol* = 'cd \$FOAM\_SOLVERS'.

Source code to solve a particular continuum mechanics problem.

### 1.2 *test*:

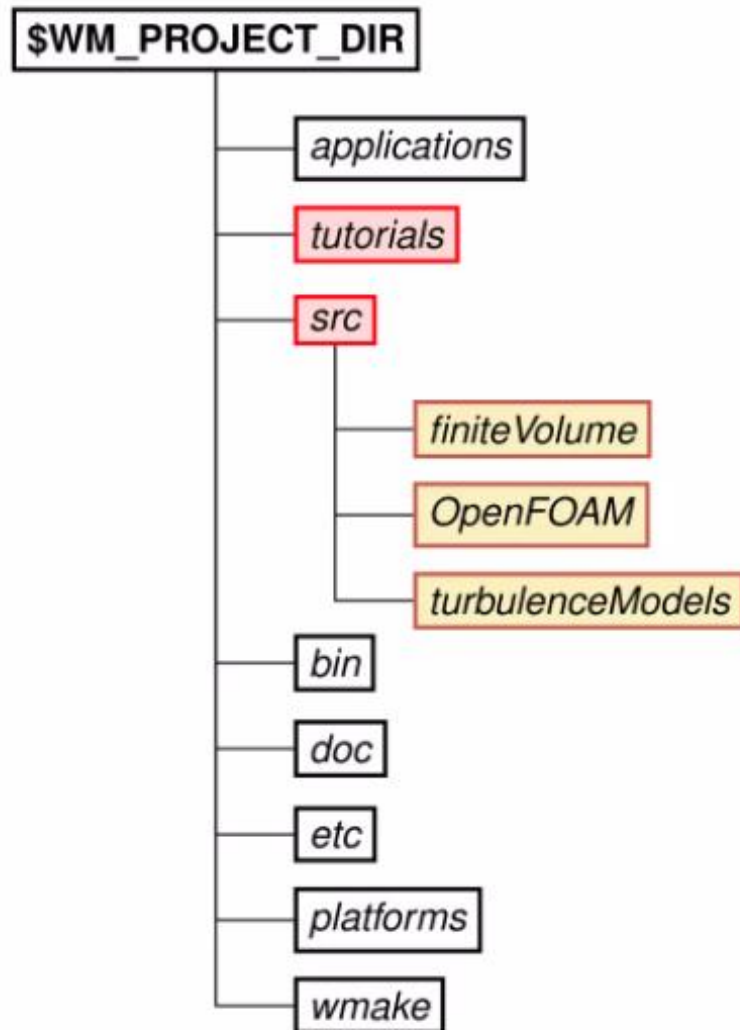
Sample codes to help understand the usage of OpenFOAM libraries.

### 1.3 *Utilities*:

alias: *util* = 'cd \$FOAM\_UTILITIES'.

Source code to perform pre- and post-processing tasks involving data manipulation and algebraic manipulations.

# OpenFOAM Structure



## 2. *tutorials*:

alias: *tut* = 'cd \$FOAM\_TUTORIALS'.

Contains tutorials that demonstrate the usage of all solvers and most of the utilities.

## 3. *src*:

alias: *src* = 'cd \$FOAM\_SRC'.

It contains several subdirectories which include the source code for all libraries. The important folders are:

### 3.1 *finiteVolume*:

alias: *foamfv* = 'cd \$FOAM\_SRC/finiteVolume'.

Includes *classes* for finite volume space/time discretisation, boundary conditions etc.

### 3.2 *OpenFOAM*:

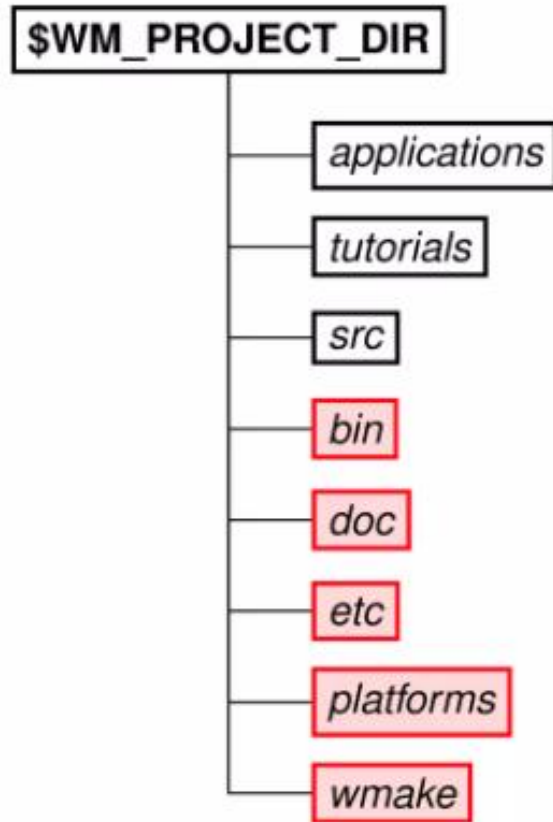
This core library includes important definitions.

### 3.3 *turbulenceModels*:

Contains libraries for turbulence models.



# OpenFOAM Structure



## 4. *bin*:

This directory contains shell scripts such as *paraFoam*, *foamLog* etc.

## 5. *doc*:

It contains all the **documentation** relevant to the version of **OpenFOAM** including:

- 5.1 User and Programmer's guides
- 5.2 Doxygen generated documentation
- 5.3 **OpenFOAM** coding style guide

## 6. *etc*:

It contains global **OpenFOAM** dictionaries and setup files.

## 7. *platforms*:

The binaries generated during the compilation of the applications and the dynamic libraries are stored here.

## 8. *wmake*:

**Compiler settings** are included in this directory including optimisation flags. It also contains *wmake*, a special make command which understands the **OpenFOAM** file structure.

# OpenFOAM Case Structure

- Basic OpenFOAM case

\$FOAM\_TUTORIALS/multiphase/interFoam/RAS/damBreak/damBreak

- This is not a comprehensive list of case files!
- Some files can be placed at different directory, i.g. blockMesh

## Time Directories

- Contains the field values and boundary conditions of specific time

## Constant directory

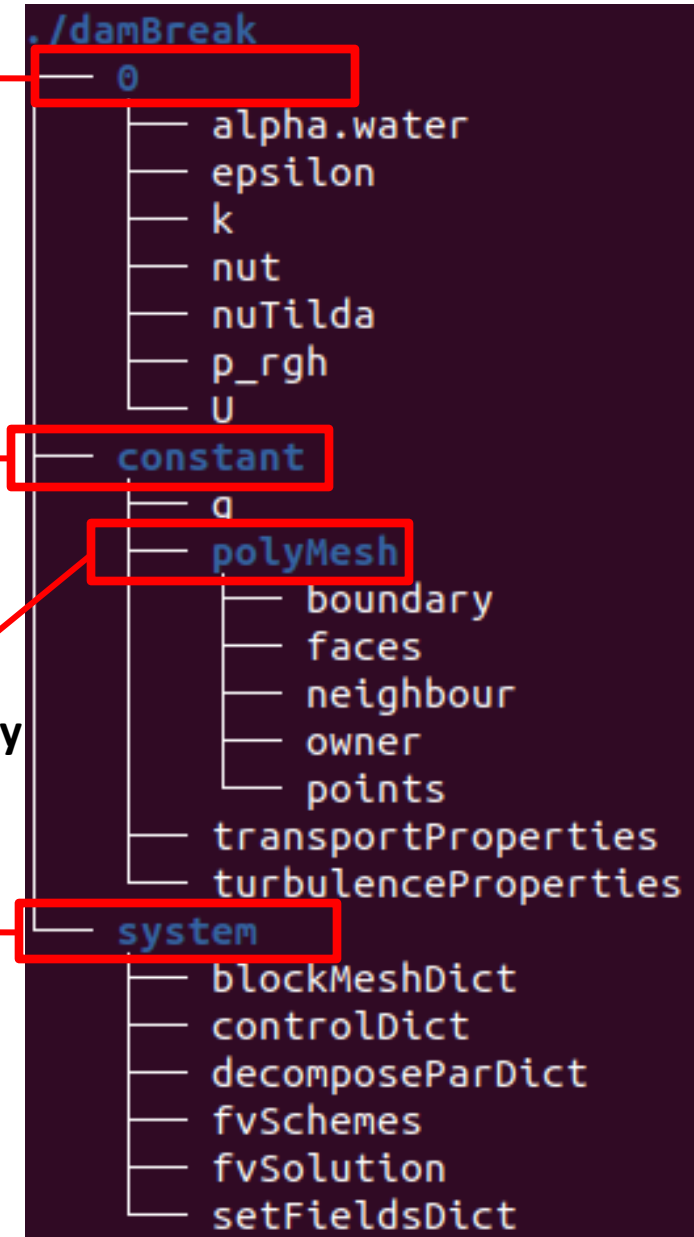
- Contains the physical properties and the mesh information

## PolyMesh sub-directory

- Contains mesh info

## System directory

- Mainly for defining the numerical schemes and their parameters.





Science and  
Technology  
Facilities Council

Scientific Computing

# Numerics of OpenFOAM



Science and  
Technology  
Facilities Council

Scientific Computing

# Governing Equation

- General conservation equation

$$\underbrace{\frac{\partial \rho \phi}{\partial t}}_{\text{Time derivative}} = - \underbrace{\nabla \cdot \rho \mathbf{u} \phi}_{\text{Convection}} + \underbrace{\nabla \cdot (D \nabla \phi)}_{\text{Diffusion}} + \underbrace{S_\phi}_{\text{Source term}}$$

- Momentum conservation equation

$$\frac{\partial \rho \mathbf{u}}{\partial t} = -\nabla \cdot \rho \mathbf{u} \mathbf{u} + \nabla \cdot (\rho \nu \nabla \mathbf{u}) + -\nabla p + S$$

- Mass Conservation Equation

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot \rho \mathbf{u}$$

# Finite Volume Method

- The domain is discretized into small cells, and the values of the cell is the integral of the function inside the cell.
- Insure flux conservation, and can be used for complex geometry and mesh refinement.
- OF uses cell-centered finite volume method.
- Integral form of the conservation equation:

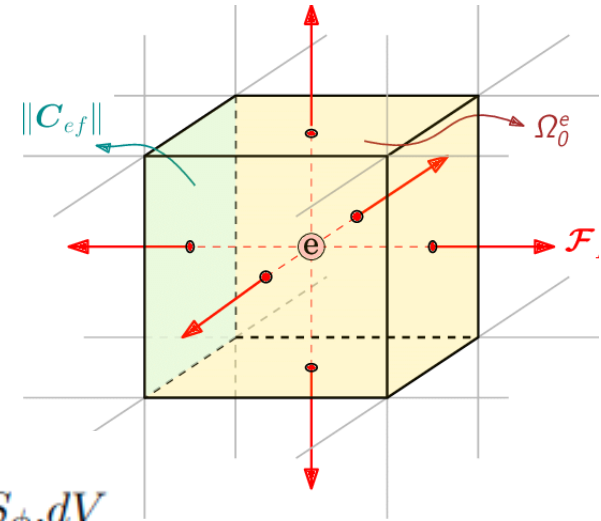
$$\int_V \frac{\partial \rho \phi}{\partial t} . dV = - \int_V \nabla \cdot \rho \mathbf{u} \phi . dV + \int_V \nabla \cdot (D \nabla \phi) . dV + \int_V S_\phi . dV$$

- Gauss's divergence theorem:

$$\int_V \nabla \cdot \mathbf{a} dV = \int_S \mathbf{a} dS$$

- Then the conservation equation cab be expressed as:

$$\int_V \frac{\partial \rho \phi}{\partial t} dV = - \int_S \rho \mathbf{u} \phi dS + \int_S (D \nabla \phi) dS + \int_V S_\phi dV$$



# Finite Volume Method

- The domain is discretized into small cells, and the values of the cell is the integral of the function inside the cell.
- Insure flux conservation, and can be used for complex geometry and mesh refinement.
- OF uses cell-centered finite volume method.
- Integral form of the conservation equation:

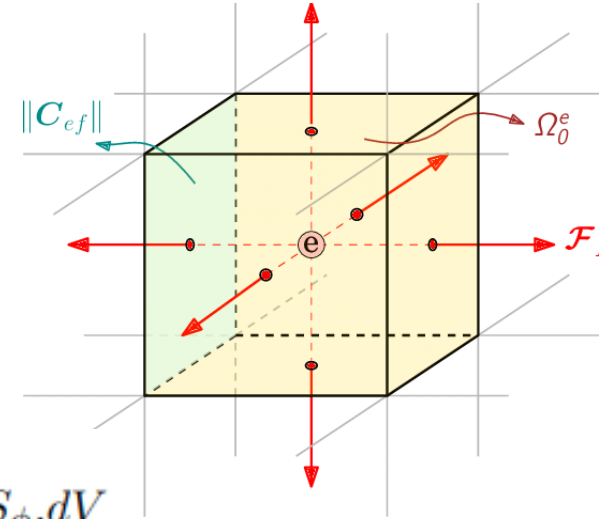
$$\int_V \frac{\partial \rho \phi}{\partial t} .dV = - \int_V \nabla \cdot \rho \mathbf{u} \phi .dV + \int_V \nabla \cdot (D \nabla \phi) .dV - \int_V S_\phi .dV$$

- Gauss's divergence theorem:

$$\int_V \nabla \cdot \mathbf{a} dV = \int_S \mathbf{a} dS$$

- Then the conservation equation cab be expressed as:

$$\int_V \frac{\partial \rho \phi}{\partial t} dV = - \int_S \rho \mathbf{u} \phi dS + \int_S (D \nabla \phi) dS + \int_V S_\phi dV$$





# Finite Volume Method: Time Derivative

$$\int_V \frac{\partial \rho \phi}{\partial t} dV = \frac{\partial}{\partial t} \int_V \rho \phi dV$$

- In Second order discretisation we assume that  $\phi$  changes linearly in space:

$$\phi(\mathbf{x}) = \phi_p + (\mathbf{x} - \mathbf{x}_p) \cdot \nabla \phi$$

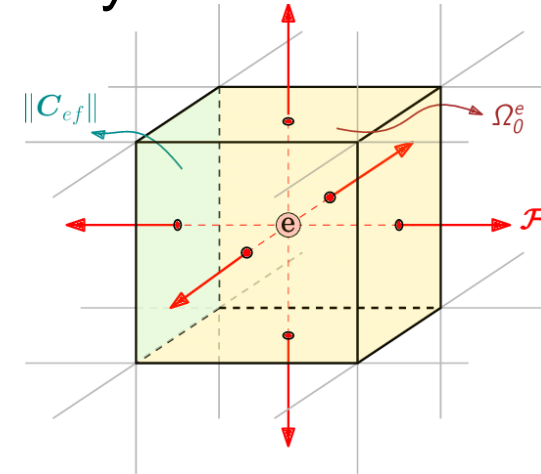
- The volume integration of  $\phi$

$$\int_V \phi(\mathbf{x}) dV = \int_V \phi_p dV + \int_V (\mathbf{x} - \mathbf{x}_p) \cdot \nabla \phi_p dV$$

- Note:  $\int_V (\mathbf{x} - \mathbf{x}_p) \cdot \nabla \phi_p dV = 0$  as  $x_p$  in the center of the cell and  $\nabla \phi_p$  is constant

$$\int_V \phi(\mathbf{x}) dV = \phi_p V$$

$$\int_V \frac{\partial \rho \phi}{\partial t} dV = \frac{\partial}{\partial t} \int_V \rho \phi dV \rightarrow \frac{\partial}{\partial t} (\rho_p \phi_p) V$$



# Finite Volume Method: Convection Term

$$\int_V \nabla \cdot \rho \mathbf{u} \phi \, dV = \int_S \rho \mathbf{u} \phi \, dS = \sum_i S(\rho \mathbf{u} \phi)_f = \sum_i F \phi_f$$

- Where  $F$  is the flux, and  $i$  is the iterator over the cell,
- and  $S$  is surface area of the face
- **Central Differencing (CD)**

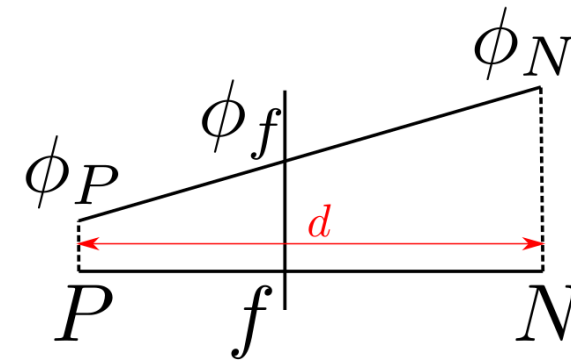
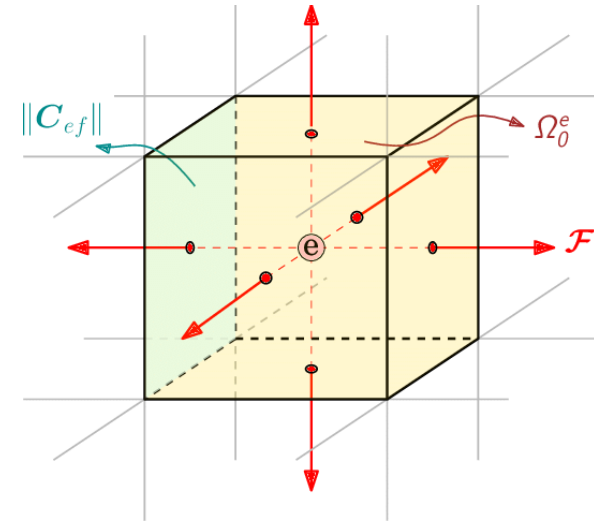
$$\phi_f = \alpha \phi_P + (1 - \alpha) \phi_N$$

$$\alpha = \frac{\overline{fN}}{d}$$

- Second order, unbounded, causes unphysical oscillations
- **Upwind differencing (UD)**

$$\phi_f = \begin{cases} \phi_P & , F \geq 0. \\ \phi_N & , F < 0. \end{cases}$$

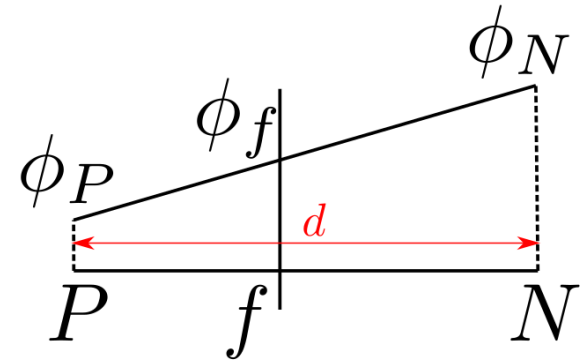
- First order, bounded faces



# Finite Volume Method: Diffusion Term

$$\int_V \nabla \cdot (D \nabla \phi) dV = \int_S D \nabla \phi dS = \sum_i S \cdot (D \nabla \phi)_f$$

$$(\nabla \phi)_f = \begin{cases} \frac{\phi_N - \phi_P}{d}, & \text{For orthogonal grid.} \\ \alpha(\nabla \phi)_P + (1 - \alpha)(\nabla \phi)_N, & \text{For non-orthogonal grid.} \end{cases}$$



- Both are second-order accurate but differ in boundness and suitability for non-orthogonal meshes

$$S \cdot (\nabla \phi)_f = \underbrace{\Delta \cdot (\nabla \phi)_f}_{\text{orthogonal contribution}} + \underbrace{\kappa \cdot (\nabla \phi)_f}_{\text{non-orthogonal contribution}}$$

$$S = \Delta + \kappa$$

$$\Delta = \begin{cases} \frac{d \cdot S}{d \cdot d} d & \text{Minimum correction approach.} \\ \frac{d}{|d|} |S| & \text{Orthogonal correction approach.} \\ \frac{d}{d \cdot S} |S|^2 & \text{Over-relaxed approach.} \end{cases}$$

# Finite Volume Method: Usage in OF

- The gradient and divergence schemes are defined in *\$case/system/fvSchemes*

```
gradSchemes
{
    default          none;  It can be any scheme, i.e. Gauss linear
    grad(p)          <optional limiter> <gradient scheme> <interpolation scheme>;
    .....
    .....
}

divSchemes
{
    default          none;
    div(phi,T)       Gauss <interpolation scheme>;
}
```

- There are different limiter, gradient and interpolations schemes.

<https://www.openfoam.com/documentation/guides/v2112/doc/guide-schemes-gradient.html>

<https://www.openfoam.com/documentation/guides/v2112/doc/guide-schemes-divergence-example.html>



Science and  
Technology  
Facilities Council

Scientific Computing

# Numerics of OpenFOAM

## Time discretisation



Science and  
Technology  
Facilities Council

Scientific Computing



# Time Integration for Transient Flows

$$\int_t^{t+\Delta t} \left( \frac{\partial \rho \phi}{\partial t} \right)_P V_P dt = \int_t^{t+\Delta t} \left[ - \sum_f F \phi_f + \sum_f DS \cdot (\nabla \phi)_f + S_\phi V_P \right] dt$$

- Backward time scheme
- Euler implicit time scheme
- Crank-Nicolson time scheme
- <https://www.openfoam.com/documentation/guides/v2112/doc/guide-schemes-time.html>



# Time Integration: Explicit

$$\int_t^{t+\Delta t} \left( \frac{\partial \rho \phi}{\partial t} \right)_P V_P dt = \int_t^{t+\Delta t} \left[ - \sum_f F \phi_f + \sum_f D S \cdot (\nabla \phi)_f + S_\phi V_P \right] dt$$
$$\left( \frac{\partial \rho \phi}{\partial t} \right)_P = \left( \frac{\rho^n \phi^n - \rho^o \phi^o}{dt} \right)_P$$

- The super script `n` and `o`, refers to values at new time `t+dt` and the previous time `t`.
- For clarity drop the density, i.e. consider incompressible flow

$$\left( \frac{\partial \phi}{\partial t} \right)_P = \left( \frac{\phi^n - \phi^o}{dt} \right)_P$$

- All the values are taken from the old time step

$$\phi_P^n = \phi_P^o + \frac{\Delta t}{V_P} \left[ - \sum_f F^o \phi_f^o + \sum_f D^o S \cdot (\nabla \phi)_f^o + S_\phi^o V_P \right]$$

- First order, limited by the Courant number

$$Co = \frac{\mathbf{u}_f \cdot d}{\Delta t}$$

# Time Integration: Euler Implicit

$$\int_t^{t+\Delta t} \left( \frac{\partial \rho \phi}{\partial t} \right)_P V_P dt = \int_t^{t+\Delta t} \left[ - \sum_f F \phi_f + \sum_f D S \cdot (\nabla \phi)_f + S_\phi V_P \right] dt$$
$$\left( \frac{\partial \phi}{\partial t} \right)_P = \left( \frac{\phi^n - \phi^o}{dt} \right)_P$$

- All the values are expressed in terms of the new time step

$$\phi_f = \alpha \phi_P^n + (1 - \alpha) \phi_N^n$$
$$S \cdot (\nabla \phi)_f = \Delta \cdot \frac{\phi_N^n - \phi_P^n}{d} + \kappa \cdot (\nabla \phi)_f^n$$

- The resulted equation in matrix form is

$$a_P \phi_P^n + \sum_N a_N \phi_N^n = R_P$$

- First order, Stable even for  $Co > 1$ , Bounded

# Time Integration: Backward Differencing

$$\int_t^{t+\Delta t} \left( \frac{\partial \rho \phi}{\partial t} \right)_P V_P dt = \int_t^{t+\Delta t} \left[ - \sum_f F \phi_f + \sum_f DS \cdot (\nabla \phi)_f + S_\phi V_P \right] dt$$

- All the values on the RHS are expressed in terms of the new time step

$$\left( \frac{\partial \phi}{\partial t} \right)_P = \cancel{\left( \frac{\phi^n - \phi^o}{\Delta t} \right)_P} \quad \frac{\partial \phi}{\partial t} = \frac{1.5\phi^n - 2\phi^o + 0.5\phi^{oo}}{\Delta t}$$

- Need to solve a system of algebraic equations
- **Properties:** Implicit, Second order, Boundedness not guaranteed, Conditionally stable

# Time Integration: Crank-Nicholson

$$\int_t^{t+\Delta t} \left( \frac{\partial \rho \phi}{\partial t} \right)_P V_P dt = \int_t^{t+\Delta t} \left[ - \sum_f F \phi_f + \sum_f D S \cdot (\nabla \phi)_f + S_\phi V_P \right] dt$$

$$\left( \frac{\partial \phi}{\partial t} \right)_P = \left( \frac{\phi^n - \phi^o}{\Delta t} \right)_P$$

- All the values are expressed in terms of the new and the old time steps

$$\int_t^{t+\Delta t} \phi dt = 0.5(\phi^n + \phi^o) \Delta t$$

$$\begin{aligned} \frac{\phi_P^n - \phi_P^o}{\Delta t} V_P &= \frac{1}{2} \left[ - \sum_f F^n \phi_f^n + \sum_f D^n S \cdot (\nabla \phi)_f^n + S_\phi^n V_P \right] \\ &+ \frac{1}{2} \left[ - \sum_f F^o \phi_f^o + \sum_f D^o S \cdot (\nabla \phi)_f^o + S_\phi^o V_P \right] \end{aligned}$$

- Need to solve a system of algebraic equations
- **Properties:** Second order, Bounded

# Time Integration: Usage

- The Time integration schemes are defined in *\$case/system/fvSchemes*

```
ddtSchemes
{
    default          Euler;
    //default        CrankNicolson < a number \Psi =[0 1]>;
}
```

- Schemes list names
  - **Euler**: transient, first order implicit, bounded.
  - **backward**: transient, second order implicit, potentially unbounded.
  - **CrankNicolson**: transient, second order implicit, bounded.

$$\Psi = \begin{cases} 1 & \text{Corresponds to pure Crank-Nicolson} \\ 0 & \text{Corresponds to pure Euler} \end{cases}$$

- <https://www.openfoam.com/documentation/guides/v2112/doc/guide-schemes-time-example.html>



Science and  
Technology  
Facilities Council

Scientific Computing

# Numerics of OpenFOAM Pressure Velocity Coupling



Science and  
Technology  
Facilities Council

Scientific Computing



# Pressure-Velocity Coupling

$$a_P U_P + \sum_N a_N U_N = R_P \quad , \text{ or } \quad MU = R$$

- With ' $N$ ' being the eight

$$R = -\nabla p + S(U)$$

$$a_P U_P = -\nabla p + \boxed{S(U_P) - \sum_N a_N U_N}$$

$$a_P U_P = -\nabla p + \boxed{H_N} \quad , \text{ or } \quad AU = H - \nabla p$$

- $A$  is a diagonal matrix

$$U = A^{-1}H - A^{-1}\nabla p$$

- Substitute in the continuity equation  $\nabla \cdot \mathbf{u} = 0$

$$\nabla \cdot A^{-1}\nabla p = \nabla \cdot (A^{-1}H)$$

# Pressure velocity Coupling

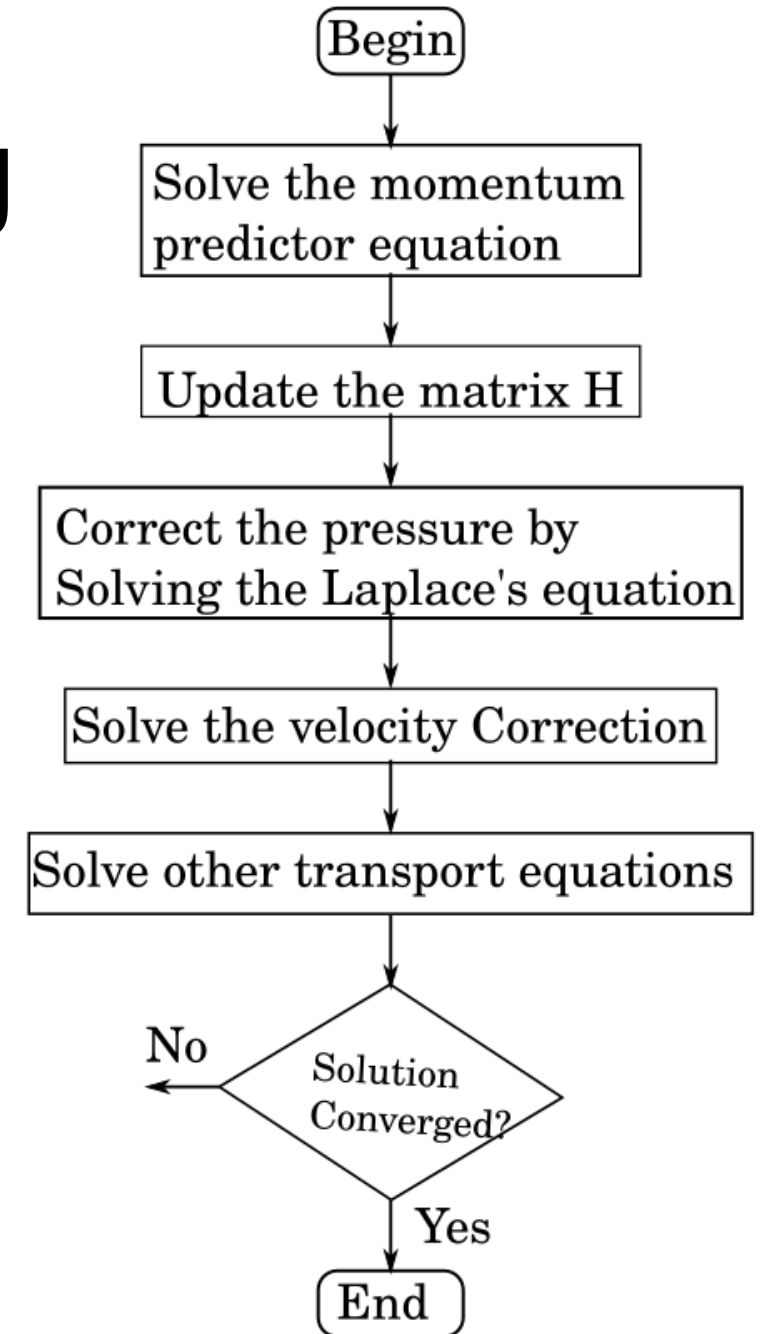
- Summary of the equation to be solved:

$$MU = -\nabla p - S(U) \quad \text{Momentum predictor}$$

$$H = AU - MU$$

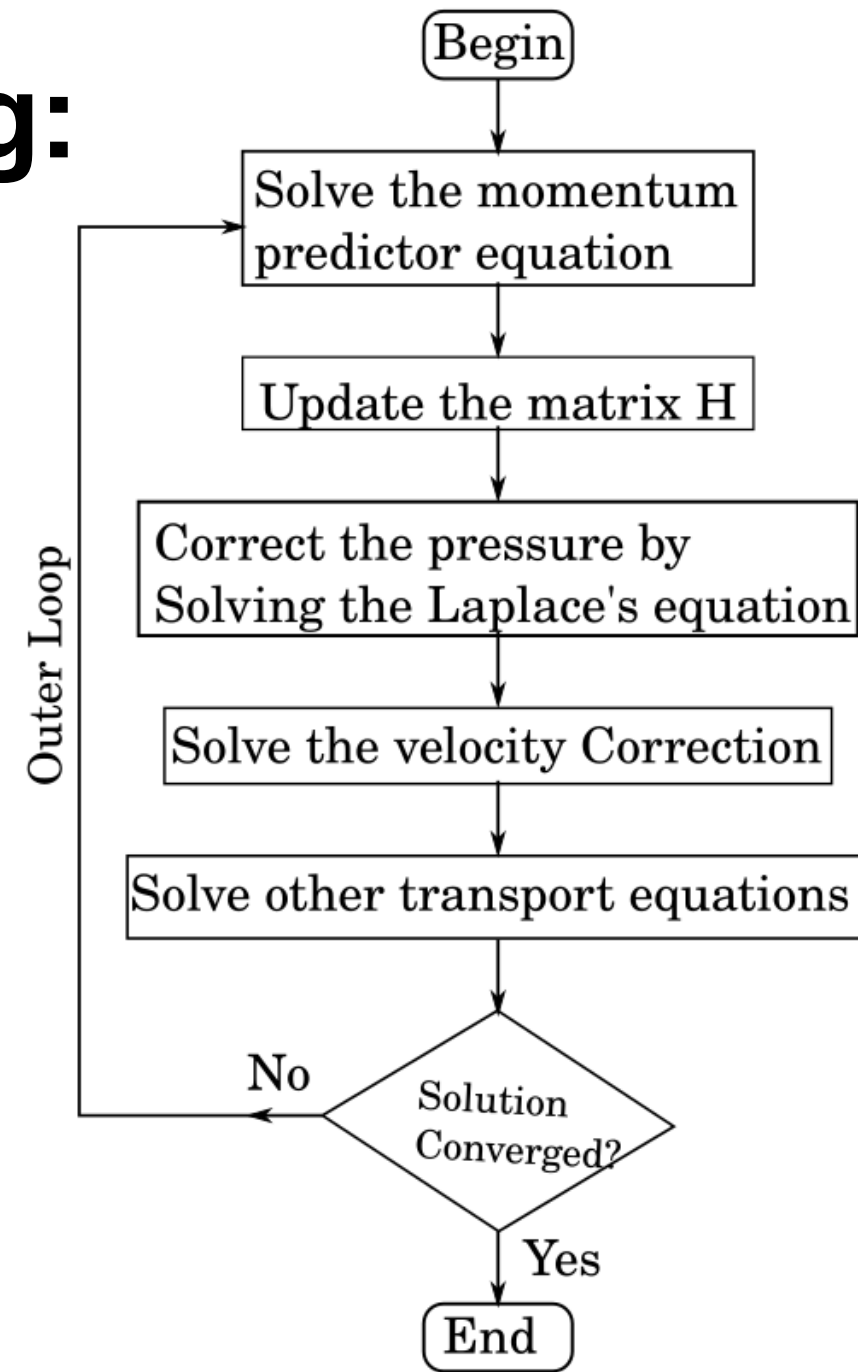
$$\nabla \cdot A^{-1} \nabla p = \nabla \cdot (A^{-1} H) \quad \text{Laplace's equation of the pressure}$$

$$U = A^{-1} H - A^{-1} \nabla p \quad \text{Velocity Corrector}$$



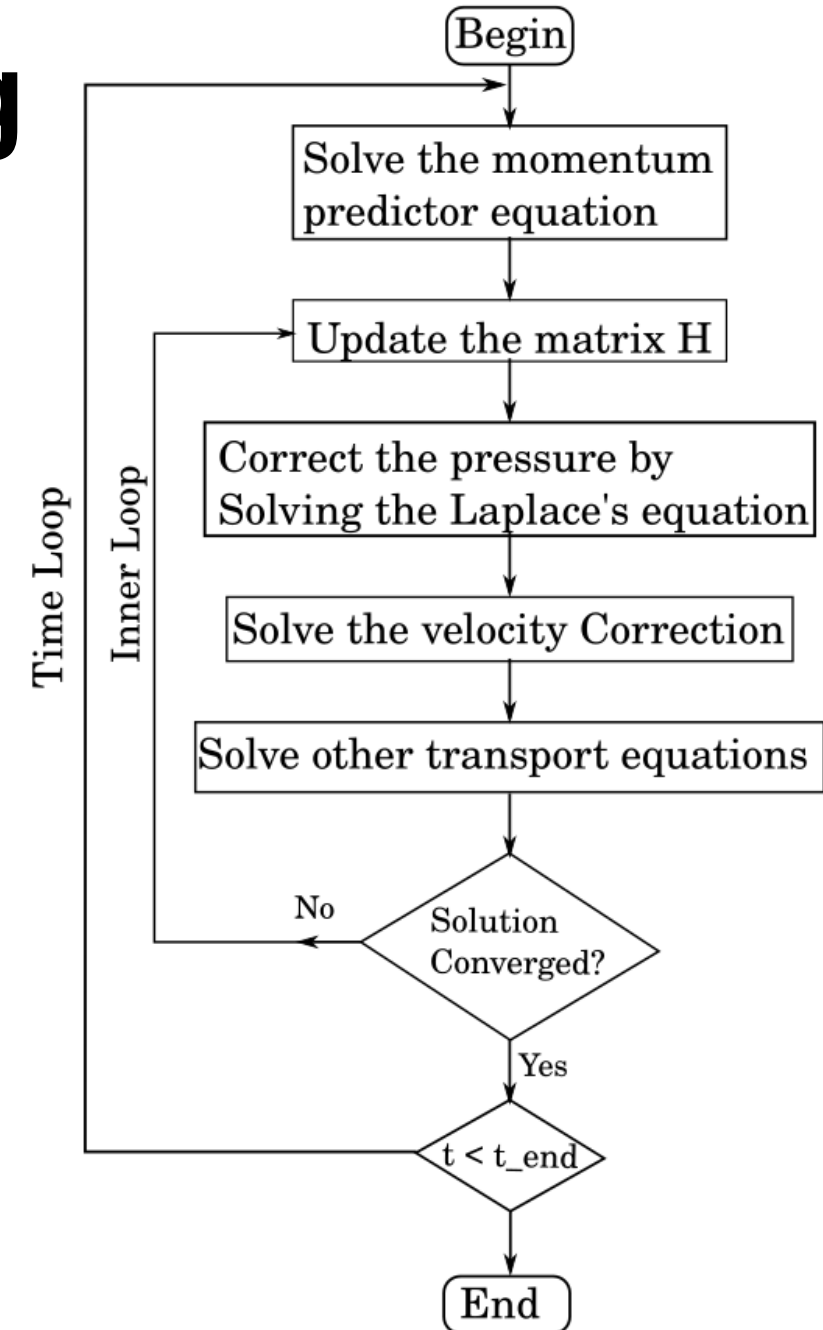
# Pressure velocity Coupling: SIMPLE

- Semi-Implicit-Method-Of-Pressure-Linked-Equations.
- Used for Steady State simulations, but needs under-relaxation.
- Stable for large Co.
- <https://www.openfoam.com/documentation/guides/v2112/doc/guide-applications-solvers-simple.html>



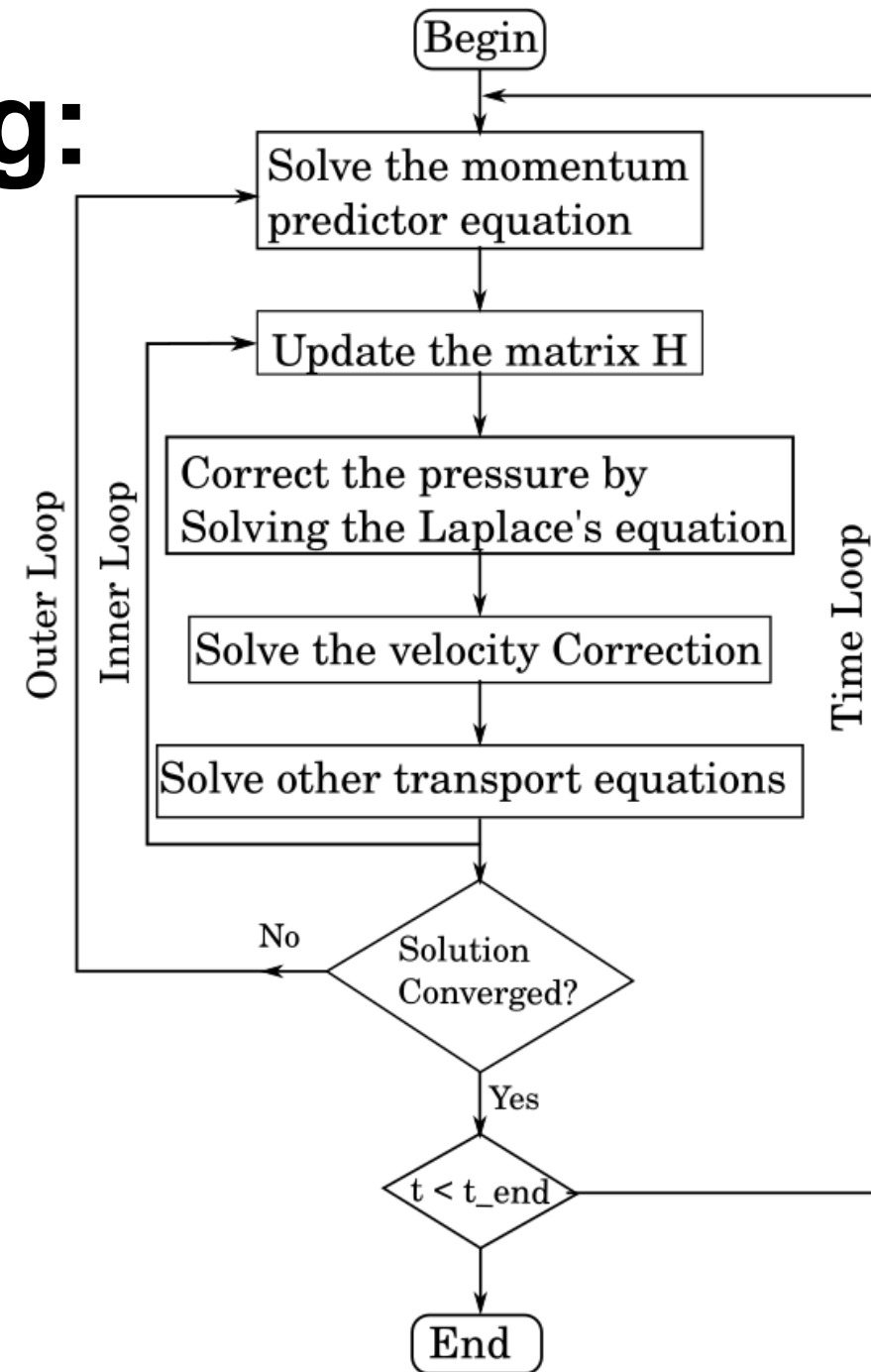
# Pressure velocity Coupling PISO

- Pressure Implicit with Splitting of Operators
- Unsteady simulation
- The momentum equation is solved once each time step
- The correction is repeated via the inner loop till convergence in each time step
- Issa, Raad I. "Solution of the implicitly discretised fluid flow equations by operator-splitting." Journal of computational physics 62.1 (1986): 40-65.



# Pressure velocity Coupling: PIMPLE

- Compines SIMPLE and PISO alogrithms
- Unsteady simulation
- The momentum equation can be solved multiple times time step (outer loop)
- The correction is repeated via the inner loop
- Allowes large  $Co$



# Pressure velocity Coupling: Usage

- Defined in *\$case/system/fvSolution*
- [https://openfoamwiki.net/index.php/OpenFOAM\\_guide/The\\_PIMPLE\\_algorithm\\_in\\_OpenFOAM](https://openfoamwiki.net/index.php/OpenFOAM_guide/The_PIMPLE_algorithm_in_OpenFOAM)
- Monitor the residuals
  - foamLog <logFile>*
  - In *\$case/system/controlDict*, add residuals function

```
functions
{
    residuals
    {
        type                solverInfo;
        libs                 (utilityFunctionObjects);
        writeResidualFields true;
        writeControl          writeTime;
        fields                (".*"); // (U, P)
    }
}
```

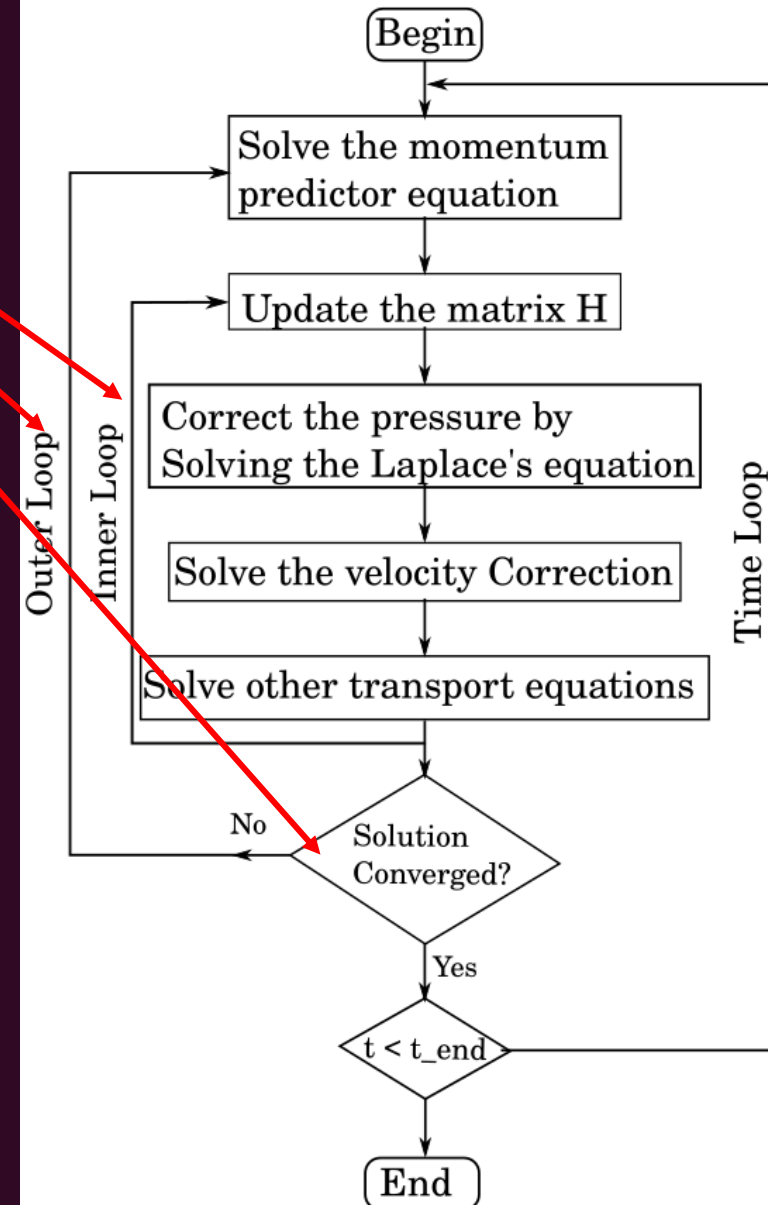
Get the residuals for all the available

Get the residuals for specific fields

```
PIMPLE
{
    nCorrectors      5;
    nOuterCorrectors 50;

    residualControl
    {
        U
        {
            tolerance 1e-5;
            relTol    0;
        }
        P
        {
            tolerance 5e-4;
            relTol    0;
        }
    }

    relaxationFactors
    {
        fields
        {
            p      0.3;
            pFinal 1;
        }
        equations
        {
            "U|k|epsilon"      0.3;
            "(U|k|epsilon)Final" 1;
        }
    }
}
```







Science and  
Technology  
Facilities Council

Scientific Computing

# Numerics of OpenFOAM

## Volume of Fluid



Science and  
Technology  
Facilities Council

Scientific Computing



# Volume of Fluid (VOF)

- Interface capturing methods use a fixed grid
- Uses a labelling function for each cell having value [0 1]

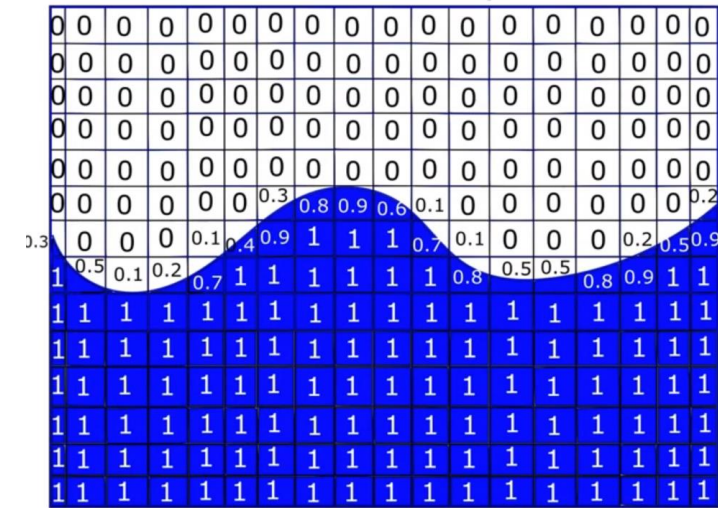
$$\alpha = \frac{V_1}{V} = \begin{cases} 1 & \text{Fluid 1} \\ 0 < \alpha < 1 & \text{Interface Cells} \\ 0 & \text{Fluid 2} \end{cases}$$

- Assume that the local properties changes linearly with the volume fraction

$$\rho = \alpha\rho_1 + (1 - \alpha)\rho_2$$

$$\mu = \alpha\mu_1 + (1 - \alpha)\mu_2$$

- The governing equations can be used with no change
  - Note: Source term to account for the surface tension can be added



Source:

[https://www.youtube.com/watch?v=7W8JqP1Le3I&ab\\_channel=AppliedComputationalFluidDynamics](https://www.youtube.com/watch?v=7W8JqP1Le3I&ab_channel=AppliedComputationalFluidDynamics)

# Volume of Fluid (VOF)

- There are two categories[1]:
  - **Algebraic Method** schemes are typically much simpler to implement, more efficient and are not restricted to structured meshes but not as accurate.
  - **Geometric methods** involving an explicit reconstruction of the interface from the volume fraction data. They usually involve complex geometric operation making their implementation cumbersome.
- OpenFOAM ESI (V1706 and later) has these two methods built-in
  - Algebraic Method using *MULES* scheme
  - Geometric Method using *isoAdvector*
- Note: isoAdvector is available as Third party solver for other versions of OpenFOAM
  - <https://github.com/isoAdvector>

[1] [http://dx.doi.org/10.17196/OS\\_CFD#YEAR\\_2017](http://dx.doi.org/10.17196/OS_CFD#YEAR_2017)

# Volume of Fluid (VOF): Algebraic Method

- To track the interface, a transport equation of the volume fraction is needed

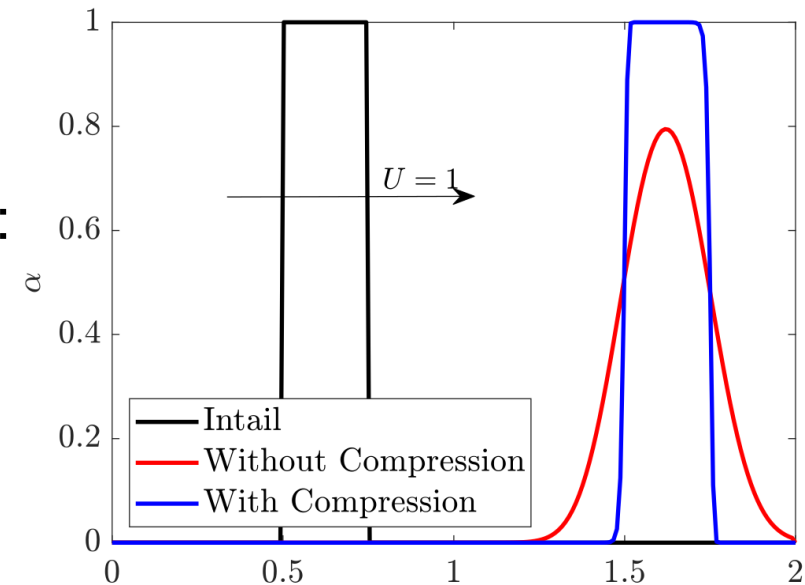
$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{u}\phi) = 0$$

- The diffusion is high leading to surface smearing
- To reduce surface smearing, a compression term is added:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{u}\phi) + \nabla \cdot (U_C \alpha(1 - \alpha)) = 0$$

$$U_C = C_\alpha |\mathbf{u}| \frac{\nabla \alpha}{|\nabla \alpha|}$$

- Where  $C_\alpha$  is user specific, 0 for no compression or 1 for strong compression
- Note: The actual implementation in OF uses two schemes
  - Upwind differencing away from the interface (reduce the computational effort)
  - High order scheme at the interface

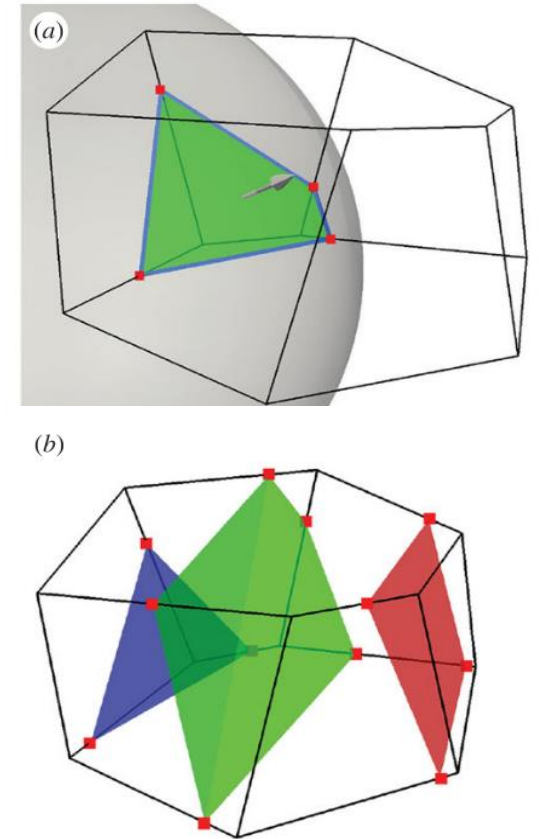


# Volume of Fluid (VOF): isoAdvectord Method

- **Geometric method**

- Reconstruct the surface of the interface using the isosurface concept of the volume fraction
- Model the motion of the constructed surface

- Olsson, E.: A description of isoAdvectord - a numerical method for improved surface sharpness in two-phase flows. In Proceedings of CFD with OpenSource Software, 2017,
- Roenby J, Bredmose H, Jasak H. A computational method for sharp interface advection. Royal Society open science. 2016 Nov 23;3(11):160405.



# Important resources

- OpenFOAM Guides
  - <https://www.openfoam.com/documentation/overview>
    - **User Guide:** Gain understanding of how OpenFOAM cases are assembled and evaluated in the OpenFOAM user guide:
    - **Tutorial Guide:** A collection of tutorials to help users get started with OpenFOAM covering a range of topics, including incompressible, compressible and multiphase flows, and stress analysis
    - **Extended Code Guide:** see how OpenFOAM operates under-the-hood. As an open source code, users can directly see how the code is written and learn how the functionality is implemented.
- Finite volume and time discretisation
  - [https://spiral.imperial.ac.uk/bitstream/10044/1/8335/1/Hrvoje\\_Jasak-1996-PhD-Thesis.pdf](https://spiral.imperial.ac.uk/bitstream/10044/1/8335/1/Hrvoje_Jasak-1996-PhD-Thesis.pdf)
- Numerics of OpenFOAM
  - <https://www.researchgate.net/publication/307546712>
- VOF
  - [http://dx.doi.org/10.17196/OS\\_CFD#YEAR\\_2017](http://dx.doi.org/10.17196/OS_CFD#YEAR_2017)



Science and  
Technology  
Facilities Council

Scientific Computing

# Questions?



Science and  
Technology  
Facilities Council

Scientific Computing

# Thank you

 [scd.stfc.ac.uk](https://scd.stfc.ac.uk)

 [@SciComp\\_STFC](https://twitter.com/SciComp_STFC)

 [STFC Scientific Computing](https://www.linkedin.com/company/stfc-scientific-computing)

 [Omar-ahmed.mahfoze@stfc.ac.uk](mailto:Omar-ahmed.mahfoze@stfc.ac.uk)