CFD Multiphysics

Analysis of numerical solvers in openFoam Extend 4.0

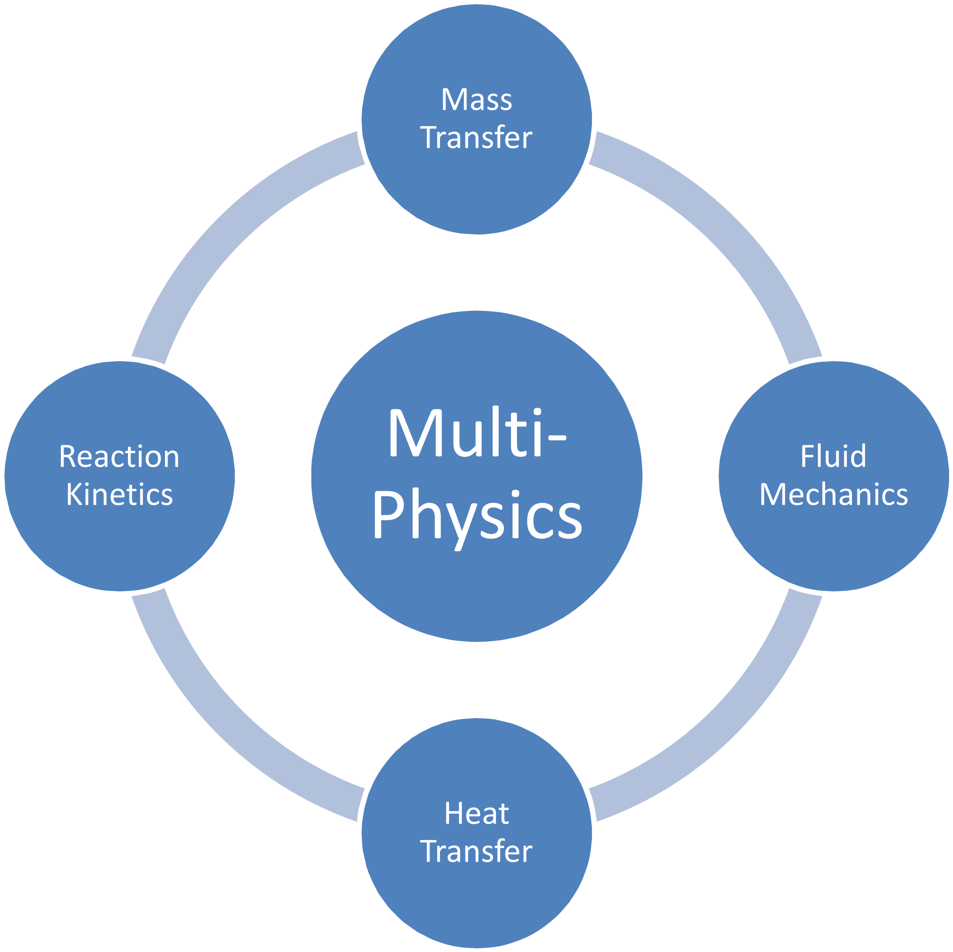
Norwegian University of Science and Technology

Agenda

- Difference between FEM and FVM
- C++ Implementation in openFOAM
- Difference between segregated and coupled solvers
- Compare convergence results of simpleFoam and pUCoupledFoam

What is CFD searching for?

Objective



Example of a Multiphysics problem

Thermal boundary layer of a system

- NSE continuity and momentum
- Temperature equation

How the problem is solved?

- Velocity field is independent of the temperature field
- The NSE is first solved and then followed by the temperature equation

Finite Element versus Finite Volume Method

Objective

- In the finite volume method, the integral form of the governing equation over the control volume is used assuming a piece-wise linear variation of the dependent variables (U,P). By integrating the flux over the CV, the fluxes are balanced across the boundaries of the individual volumes.

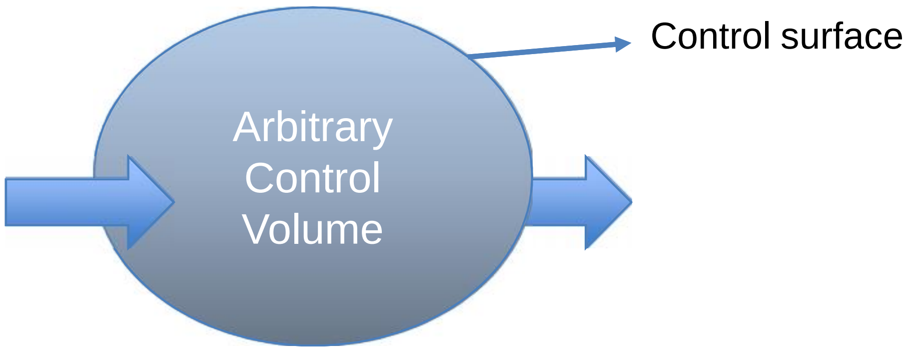
- In the finite element method, Galerkin's method of weighted residuals is generally used. In this method, the governing partial differential equations are integrated over the control volume after having been multiplied by a weight function.

- In the finite difference method, the partial derivatives are replaced with a series expansion representation, usually a Taylor series. The series is truncated usually after 1 or 2 terms.

Finite Volume Method

FVM

Rate of increase in quantity = flux of quantity in - flux of quantity out + source



Conservative equation in integral form

$$\frac{\partial}{\partial t}\int_V U\, dV = -\oint_S \phi \cdot dS + \int_V Q\, dV$$

V: volume
S: surface boundary
U: conserved quantity
phi: flux
Q: source

Finite Element versus Finite Volume Method

Comparison

Method	Advantages	Disadvantages
Finite Element	• Natural boundary conditions (for fluxes) • Master element formulation • Any shaped geometry can be modeled with the same effort	• More mathematics involved - less physical significance
Finite Volume and Finite Difference	• Fluxes have more physical significance	• Irregular geometries require far more effort

Segregated solver

Process

1. Define the momentum equation
2. Apply the under-relaxation factor
3. Solve the momentum equation
4. With the outputs, update the boundary conditions for pressure
5. Compute each element in the A and U matrix. Store the U for use in the underrelaxation
6. Interpolate for U as each face of the mesh, where U is found from the discretized momentum equation
7. Apply the pressure correction
8. Compute the continuity errors
9. Apply the momentum correction
10. Apply turbulence correction

Coupled solver

Process

1. Initialize the block matrix system Up
2. Find the explicit discretization of the divergence of the field
3. Define the momentum equation
4. The momentum equation is set up with the implicit terms with the underrelaxation factor applied.
5. The momentum is stored in Up.
6. The pressure parts of the continuity equation are set up and stored in Up.
7. Solve the block coupled matrix system Up
8. The solution is transferred to a separated field where the boundary conditions are updated.
9. Apply turbulence correction
10. Repeat until convergence is achieved.

Block coupled approach

Block coupled approach

Segregated approach

Cannot be solved for coupled systems!

$$\mathbf{A}(y)x = a$$
$$\mathbf{B}(x)y = b$$

$$\begin{bmatrix} \mathbf{A}(y) & 0 \\ 0 & \mathbf{B}(x) \end{bmatrix} \begin{bmatrix} x \\ y \end{bmatrix} = \begin{bmatrix} a \\ b \end{bmatrix}$$

For block coupled systems solver in openfoam Extend (implicit method):
`blockLduMatrix`

$$\begin{bmatrix} \mathbf{A}' & \mathbf{A}_y \\ \mathbf{B}_x & \mathbf{B}' \end{bmatrix} \begin{bmatrix} x \\ y \end{bmatrix} = \begin{bmatrix} a \\ b \end{bmatrix}$$

Introduce off-diagonal terms. A' is the term removed of the y dependence. Similarly, for B'.

$$\mathbf{C}z = c$$

$$\mathbf{C} = C_{i,j} = \begin{bmatrix} c_{i,j}^{a,a} & c_{i,j}^{a,b} \\ c_{i,j}^{b,a} & c_{i,j}^{b,b} \end{bmatrix}$$

$$c = c_i = \begin{bmatrix} a_i & b_i \end{bmatrix}^T$$

$$z = z_i = \begin{bmatrix} x_i & y_i \end{bmatrix}^T$$

Solution

Example of a block coupled system: NS momentum equation

$$\nabla \cdot (\mathbf{UU}) - \nabla(\nu \nabla \mathbf{U}) = -\frac{1}{\rho}\nabla p$$

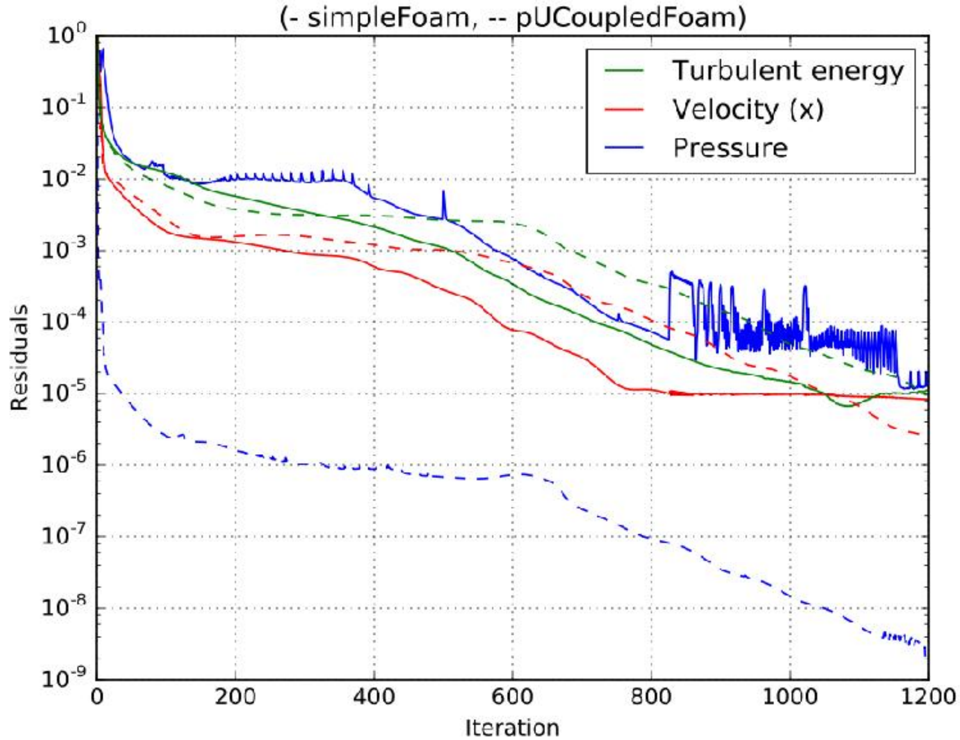
Relaxation Factor and Convergence criteria

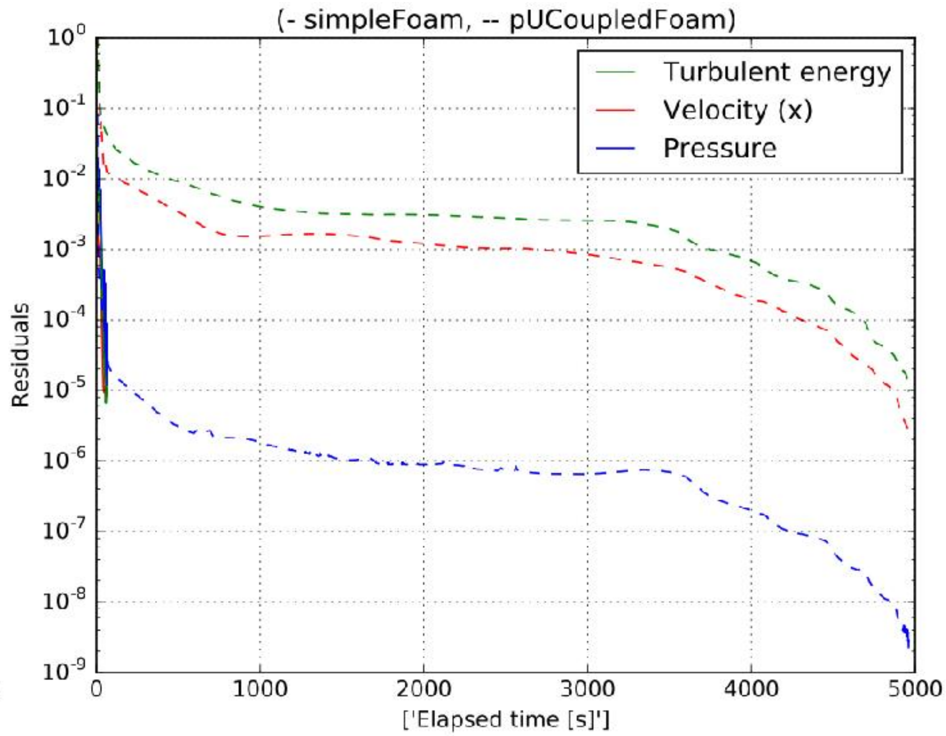
Parameters

- Absolute tolerance: Minimum residual value to achieve at the end of all the iterations.
- Relative tolerance: Relative to the initial residual. When the current residual is less than this value multiplied by the relative tolerance, the iteration stops.
- Maximum number of iterations: Upper limit of iterations regardless whether the convergence criteria is met.

pUCoupled Foam applied to pitzDaily

Convergence



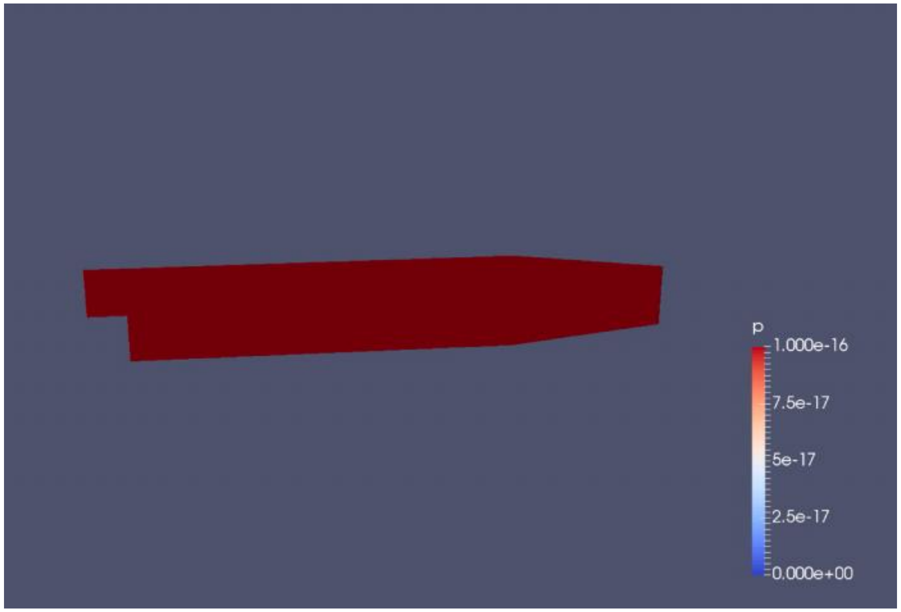


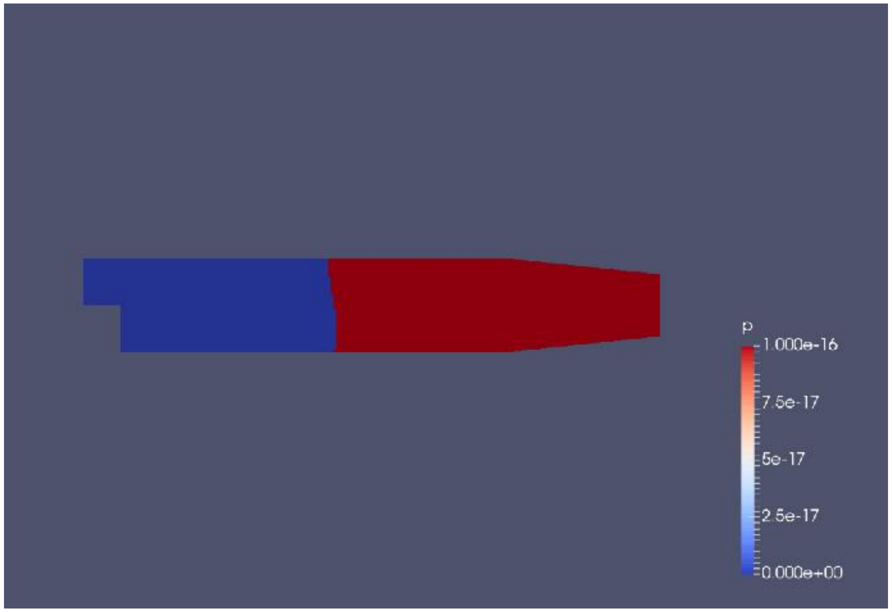
It is immediately obvious that the pUCoupledFoam did not yield any benefit over simpleFoam. pUCoupledFoam took much longer to yield the similar results as simpleFoam.

NTNU

Steady State pressure profile for *pUCoupledFoam* applied to pitzDaily

Pressure profile



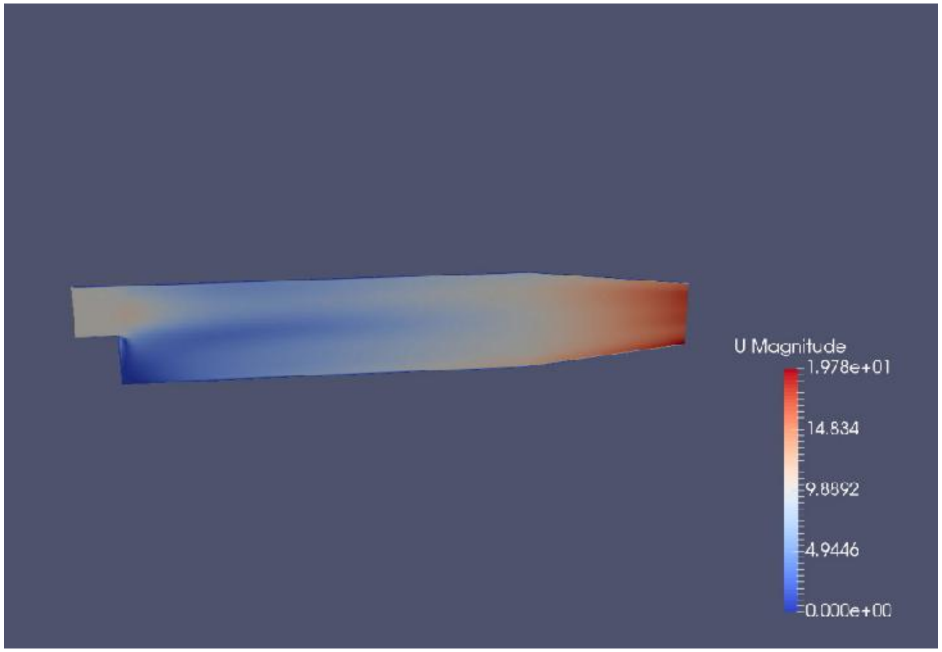


Steady state pressure profile with GMRES max iteration = 10

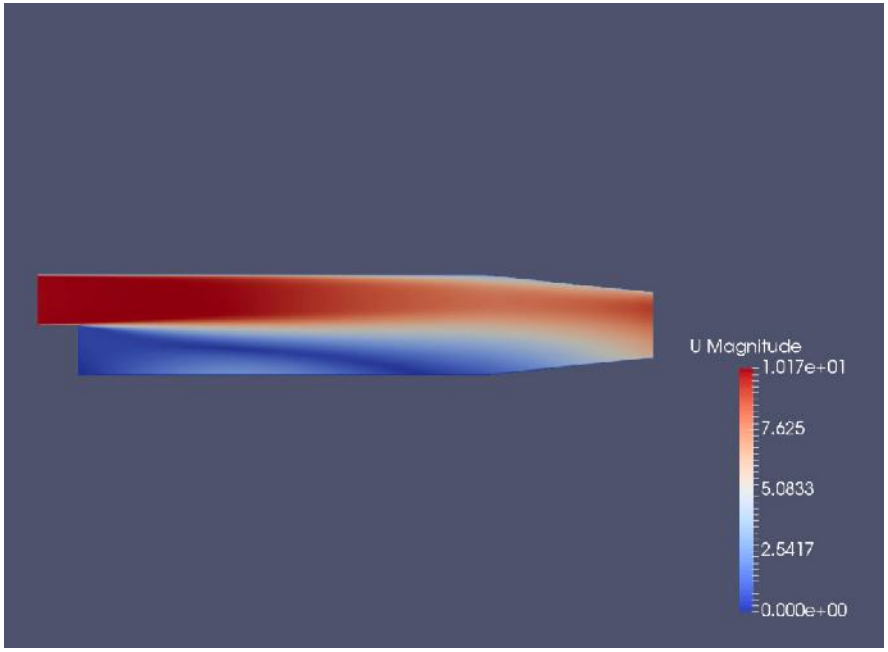
Steady state pressure profile with GMRES max iteration = 300

Steady State velocity profile for *pUCoupledFoam* applied to pitzDaily

Velocity profile



Steady state velocity profile with GMRES max iteration = 10



Steady state velocity profile with GMRES max iteration = 300