Bauman Moscow State Technical University

Solving the coupled thermo-hydrodynamical problem with the PFEM-2 method using the open-source software

Viktoriia Bondarchuk, Andrey Popov, Nikolay Tverdokhleb

Moscow — 2019

Complexity of simulation of convection

In case of convection-dominated problems finding the solution of the system of fluid dynamics equations

$$\begin{cases} \dfrac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = -\dfrac{\nabla p}{\rho} + \nu \Delta \boldsymbol{u} + \boldsymbol{f}, \\ \nabla \cdot \boldsymbol{u} = 0, \end{cases}$$

with conventional methods of CFD is complicated due to complexity of approximation of the convection term. Particle methods allow to simulate advection in a natural way using a finite set of particles.

Various particle methods

- Smooth particle hydrodynamics (SPH)
- Particle-in-cell (PIC)
- Material Point Method (MPM)
- PFEM (Particle Finite Element Method)

PFEM and PFEM-2 methods

Particles in PFEM are immaterial and carry with them no associated mass. Main purpose of their use is the simulation of convection of different variables (p, $\boldsymbol{u}$, etc.).

PFEM

A purely lagrangian method that uses a mesh which is constructed on particles as its nodes. The mesh is moving and needs to be rebuilt after each time step.

PFEM-2 (PFEM, 2nd generation)

A hybrid eulerian–lagrangian method[a] that uses a fixed mesh (no need for constant re-meshing).

[a] Idelsohn S.R., Nigro N.M., Gimenez J.M., Rossi R., Marti J.M. A fast and accurate method to solve the incompressible Navier – Stokes equations. Engineering Computations, 2013, vol. 30, issue 2, pp. 197-222. DOI: 10.1108/02644401311304854

Basic idea of the PFEM-2 method

The idea of the method

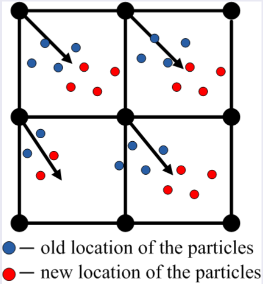
Split of the original problem into two (by physical processes):

- convection simulation using a set of particles;
- account for the influence of viscosity, pressure gradient and external body forces by solving the problem using finite element method on a fixed mesh.

Applications

- sumulation of multiphase flows;
- free surface problems;
- fluid-structure interaction problems;
- modeling of high-Reynolds number flows.

Convection of particles

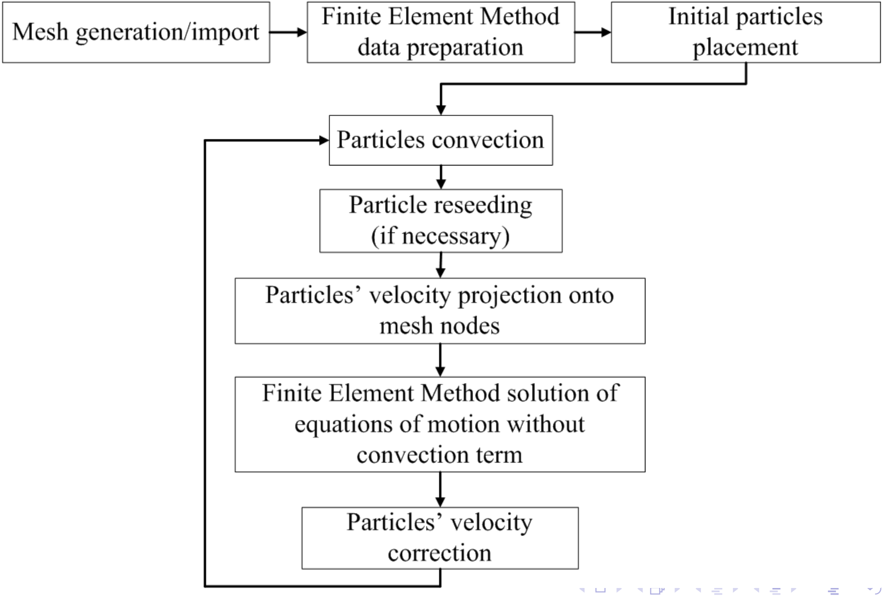


The PFEM-2 method

Advantages

- The elimination of the convection term from the FEM problem (solved on the fixed grid) leads to the possibility of using a coarser mesh and a larger time step without loss of stability (in cases when the convection is dominating);
- the problem of particles' convection requires a small time step, however all particles move independently, which allows for the effective parallelization;
- particles can carry different variables other than velocity — phase marker, temperature, etc;
- simplification of the task of finding the interface for multiphase flows.

Solution algorithm of the PFEM-2 method



Particle stage

- Particles' <u>convection</u> — given the velocity field at the mesh nodes, the particles are transferred along the velocity field streamlines using its finite element reconstruction and the explicit Euler (or Runge–Kutta) method. This process is split into several sub-steps in order to provide $CFL \leqslant 0.10 \ldots 0.15$ (whereas the overall step size can be rather high: $CFL \geqslant 1$ is allowed);
- particles' velocity <u>projection</u> — velocity field at the mesh nodes is reset to zero and recalculated using the velocities of the particles' in surrounding mesh cells;
- *FEM solution on a fixed mesh*;
- particles' velocity <u>correction</u> — the velocity of each particle is adjusted by the increment of the velocity field at the mesh nodes (compared to the previous time step), interpolation technique is used.

Mesh stage

The Navier – Stokes equations (omitting convective term) and the incompressibility equation are solved:

$$\begin{cases} \dfrac{\partial \boldsymbol{u}}{\partial t} = -\dfrac{\nabla p}{\rho} + \nu \Delta \boldsymbol{u} + \boldsymbol{f}, \\ \nabla \cdot \boldsymbol{u} = 0, \end{cases}$$

Implicit scheme is used, both monolithic and coupled strategies (including fractional step approach) can be used for calculation of new values of $\boldsymbol{u}$ and p.

1. $$\frac{\boldsymbol{u}^{n+1/2} - \boldsymbol{u}^n}{\tau} = -\frac{1}{\rho}\nabla p^n + \nu \Delta \boldsymbol{u}^{n+1/2} + \boldsymbol{f},$$
2. $$\Delta p^{n+1} = \Delta p^n + \frac{\rho}{\tau}\nabla \cdot \boldsymbol{u}^{n+1/2},$$
3. $$\frac{\boldsymbol{u}^{n+1} - \boldsymbol{u}^{n+1/2}}{\tau} = -\frac{1}{\rho}\left(\nabla p^{n+1} - \nabla p^n\right).$$

The coupled thermo-hydrodynamical problem

The Boussinesque approximation is used to simulate the free convection process:

$$\rho(T) = \rho_0 - \rho_0 \beta (T - T_0).$$

The system of equations includes the Navier – Stokes equations, the incompressibility equation and the heat transfer equation:

$$\begin{cases} \rho_0 \left(\dfrac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla) \boldsymbol{u} \right) = -\nabla p + \mu \Delta \boldsymbol{u} + \rho_0 \boldsymbol{g} - \rho_0 \beta (T - T_0) \boldsymbol{g} \\ \nabla \cdot \boldsymbol{u} = 0, \\ c \rho_0 \left(\dfrac{\partial T}{\partial t} + (\boldsymbol{u} \cdot \nabla) T \right) = \lambda \Delta T. \end{cases}$$

The temperature is a variable at the mesh nodes, particles convect temperature with them, which allows for the elimination of the convective term.

Modification of PFEM-2

Particle stage

- Particles convection — unchanged;
- particles' temperature projection — temperature field at the mesh nodes is reset and recalculated using the temperature carried by particles in the surrounding mesh cells;
- particles' temperature correction — the temperature of each particle is adjusted by the increment of the temperature field at the mesh nodes.

Mesh stage

To determine the temperature field T at the mesh nodes the heat transfer equation without the convective term is solved

$$c\rho_0 \frac{T^{n+1} - T^n}{\tau} = \lambda \Delta T^{n+1}.$$

Software implementation

Preprocessor

Mesh construction using **Salome** with subsequent import of a UNV-file (as well as built-in methods for simple cases)

Solver

deal.II[a] (Differential Equations Analysis Library) — an open-source C++ program library developed for the solution of PDE using the finite element method

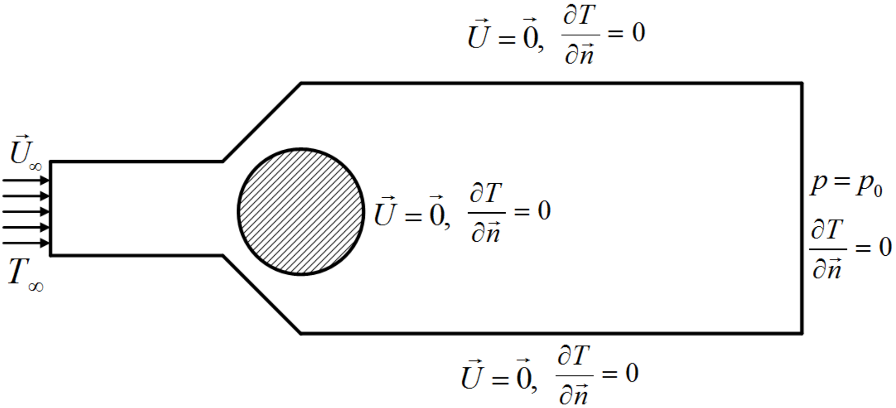
[a] The deal.II Finite Element Library. Home page. URL: http://www.dealii.org/

Postprocessor

Visualisation of the results (both particles and mesh) using **ParaView** (VTK format)

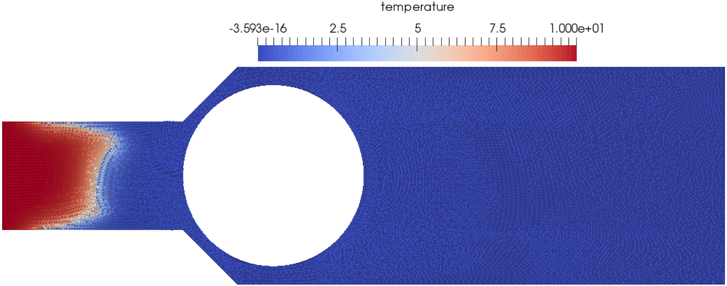
Test case: 2D flow in an expanding channel

Simulation of heat transfer in an incompressible fluid flowing through an expanding channel with a cylindrical obstacle.

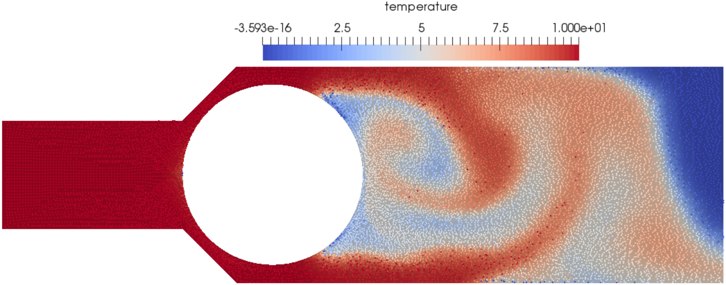


Test case: 2D flow in an expanding channel

Using the implemented algorithm the following results are obtained.



The temperature field at $t = 0.5$ sec.



The temperature field at $t = 10.0$ sec.