On OpenFOAM Efficiency for Solving Flow-Structure Interaction Problems

Kseniia S. Kuzmina, Ilia K. Marchevsky

Bauman Moscow State Technical University
Applied mathematics dep.

Guimarães
June 27th, 2016

Numerical Methods for FSI-problems

FSI coupled problems

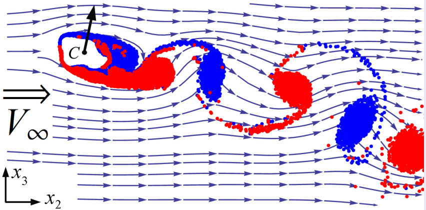
- Movable bodies
- Deformable bodies

Variable flow region

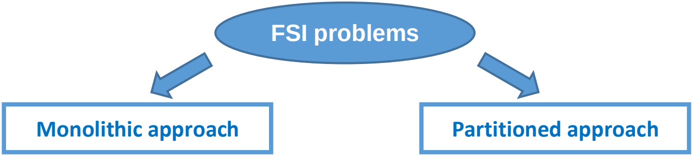
- How to construct mesh?
- How to satisfy BC?

Main assumptions

- 2D flows are considered
- Flow is viscous incompressible
- Airfoils are heavy ($\rho_0/\rho \gg 1$)
  ρ_0 — airfoil's average density
  ρ — density of the flow



Different approaches for solving FSI problems



FSI problems

Monolithic approach

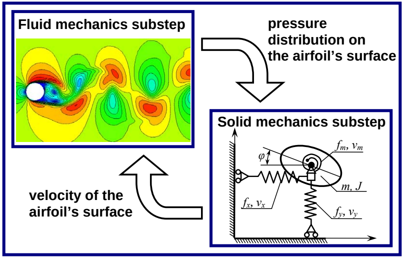
Partitioned approach

Monolithic approach

- Treats coupled fluid and structure equations simultaneously
- System is in general nonlinear, solution involves Newton's method
- **Advantages:**
  - high accuracy & stability
- **Disadvantages:**
  - expensive computation of derivatives (Jacobian matrix)
  - loss of software modularity due to the simultaneous solution of fluid and structure

1. Michler C., Hulshoff S.J., van Brummelen E.H., de Borst R. A monolithic approach to fluid-structure interaction // *Computers & Fluids*. 2004. Vol. 33, Is. 5–6. P. 839–848

Partitioned approach



Fluid mechanics substep

pressure distribution on the airfoil's surface

Solid mechanics substep

velocity of the airfoil's surface

Basic ideas

- Systems spatially decomposed into partitions
- Solution is separately advanced in time over each partition
- Partitions interact on their interface
- Interaction by transmission and synchronization of coupled state variables

Advantages & Disadvantages

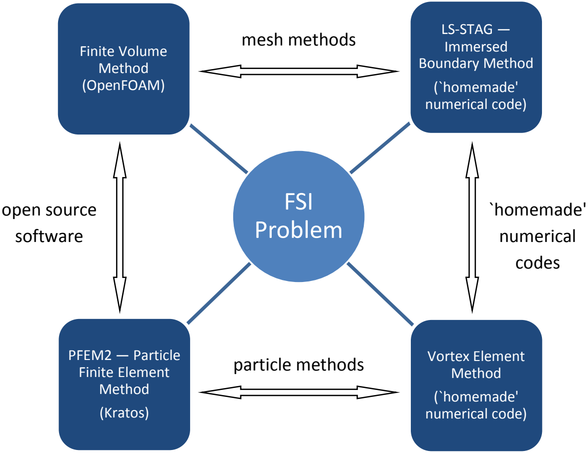
Advantages:

- customization
- independent modeling
- software reuse
- modularity

Disadvantages:

- requires careful formulation and implementation to avoid serious degradation in stability and accuracy
- parallel implementations are error-prone

Suitable Numerical Methods



Finite Volume Method (OpenFOAM)

mesh methods

LS-STAG — Immersed Boundary Method

(`homemade' numerical code)

open source software

FSI Problem

`homemade' numerical codes

PFEM2 — Particle Finite Element Method

(Kratos)

particle methods

Vortex Element Method

(`homemade' numerical code)

Governing Equations

Fluid mechanics

Continuity & Navier — Stokes equations:

$$\nabla\cdot\vec{V}=0,\qquad \frac{\partial\vec{V}}{\partial t}+(\vec{V}\cdot\nabla)\vec{V}=-\nabla p+\frac{1}{\mathrm{Re}}\Delta\vec{V}.$$

Boundary conditions:

$$\vec{V}\Big|_{\mathrm{inlet}}=\vec{V}_\infty,\qquad \frac{\partial\vec{V}}{\partial\vec{n}}\Big|_{\mathrm{outlet}}=0,\qquad \frac{\partial\vec{V}}{\partial\vec{n}}\Big|_{\mathrm{inlet\&outlet}}=0,$$

$$\vec{V}\Big|_{\mathrm{airfoil}}=\vec{V}^{\,\mathrm{ib}},\qquad \frac{\partial p}{\partial\vec{n}}\Big|_{\mathrm{airfoil}}=0.$$

$\vec{V}$ — velocity in the flow;

p — pressure;

$\vec{V}^{\mathrm{ib}}$ — velocity of the immersed boundary

Re — Reynolds number.

Governing Equations

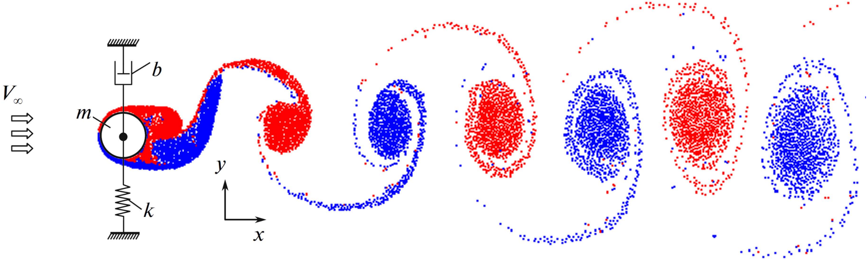
Solid mechanics — Circular airfoil oscillations across the stream

Constrain is linear viscoelastic (Kelvin — Voigt-type)

$$m\ddot{y}_* + b\dot{y}_* + ky_* = F_y.$$

y_* — deviation from the equilibrium, F_y — lift force,

m — airfoil's mass, b — damping factor, k — constraint's elasticity.



Dimensionless parameters

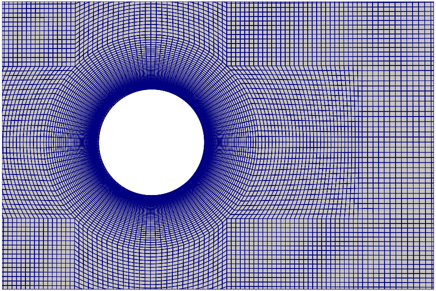
Reynolds number	Re	150
Flow velocity	V_∞	3.0
Airfoil's mass	m	39.15
Damping factor	b	0.731

Dimensionless natural frequency:

$$\mathrm{St}_\omega = \frac{1}{2\pi}\sqrt{\frac{k}{m}}\frac{D}{V_\infty} = 0.160\ldots0.220.$$

OpenFOAM Implementation

- `pimpleDyMFoam` — solver for unsteady incompressible flow and movable mesh, use PIMPLE (PISO+SIMPLE) algorithm;
- `weaklyCoupledFsi` — functionObject which allows to solve FSI-problems by using pimpleDyMFoam (tomorrow training course!);
- automatic time step: the CFL (Courant number) not more than 0.5.



- block-structured non-uniform mesh
- 114,000 cells
- 120 cells on cylinder surface

Numerical Experiment (OpenFOAM)

Velocity and vorticity distributions in the flow after circular airfoil

Wind resonance, $\mathrm{St} = 0.19$, $\mathrm{Re} = 150$

Basic Ideas of Vortex Element Method

- Purely Lagrangian meshless method.
- Vorticity $\vec{\Omega}(\vec{r}, t) = \nabla \times \vec{V}(\vec{r}, t)$ — primary compute variable.
- Navier — Stokes equations in Helmoltz form:

$$\frac{\partial \vec{\Omega}}{\partial t} + \nabla \times (\vec{\Omega} \times (\vec{V} + \vec{W})) = 0, \quad \vec{W}(\vec{r}, t) = -\nu \frac{\nabla \Omega}{\Omega}$$ — diffusive velocity[1].

- Airfoil influence is simulated by attached vortex and source layers and free vortex layer placed on its surface.
- Using Generalized Biot — Savart law we can find velocity field from vorticity field and vortex and source layers.

---

[1] Dynnikova, G.Ya. Lagrangian Approach to Solving the Time-Dependent Navier — Stokes Equations. *Dokl. Phys.* 49, 648–652 (2004).

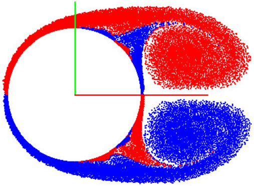
Vortex Elements

Vortex wake simulation

Vorticity distribution in the flow is simulated by large number of separate vortex elements (VE)

$$\Omega(\vec{r}) = \sum_{i=1}^{n} \Gamma_i \delta(\vec{r} - \vec{r}_i),$$

Γ_i — circulation of the VEs, $\vec{r}_i$ — their positions.



Vortex elements movement

Movement equation: $\dfrac{D\vec{\Omega}}{Dt} = 0 \quad \Leftrightarrow \quad \begin{cases} \Gamma_i = \mathrm{const}, \\ \dfrac{d\vec{r}_i}{dt} = \vec{V}(\vec{r}_i) + \vec{W}(\vec{r}_i), \quad i = 1, \dots, \end{cases}$

$$\vec{V}(\vec{r}_i) = \sum_{\substack{j=1 \\ j \neq i}}^{n} \underbrace{\frac{\Gamma_j}{2\pi} \frac{\vec{k} \times (\vec{r}_i - \vec{r}_j)}{|\vec{r}_i - \vec{r}_j|^2}}_{\vec{v}_{ij}} + \vec{V}_{\gamma}^{\mathrm{att}} + \vec{V}_{q}^{\mathrm{att}} + \vec{V}_{\infty},$$

Vortex Element Method — Examples

Blasius model problem: Thin plate in viscous flow ($\mathrm{Re} = 1000$)

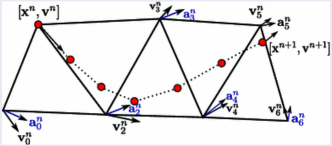
Flow around circular cylinder ($\mathrm{Re} = 1000$)

Particle Finite Element Method (PFEM-2)

- Navier — Stokes equations are non-linear; they can be written down in ‘linear’ form if Lagrangian approach is implemented:

$$\nabla \cdot \vec{V} = 0, \qquad \underbrace{\frac{\partial \vec{V}}{\partial t} + (\vec{V} \cdot \nabla)\vec{V}}_{\frac{D\vec{V}}{Dt}} = -\nabla p + \frac{1}{\mathrm{Re}} \Delta \vec{V}.$$

- Convective term: particles move in the known velocity field by using explicit (Runge — Kutta) numerical scheme with $\mathrm{CFL} \approx 0.1 \ldots 0.2$



- Diffusive and pressure term: Traditional finite element approach, 1-st order shape functions, implicit scheme, CFL up to 10.

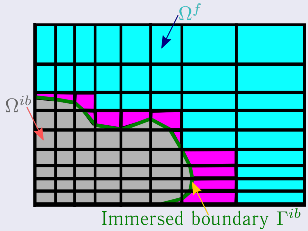
1. Juan M. Gimenez, Norberto M. Nigro, Sergio R. Idelsohn. Evaluating the performance of the particle finite element method in parallel architectures // Comp. Part. Mech. (2014) № 1. P. 103–116.

Particle Finite Element Method (PFEM-2)

Flow around circular cylinder ($\mathrm{Re} = 1000$)

Immersed Boundary (IB) methods

IB methods



Computational domain: $\Omega = \Omega^f \cup \Omega^{ib}$

Solid cells $\in \Omega^{ib} \Rightarrow$ No discretization

Fluid Cells $\in \Omega^f \Rightarrow$ Usual Cartesian discretization

«Cut-cells» $\in \Omega^f \Rightarrow$ Special IB treatment on Γ^{ib}

Features of IB-methods

- Cartesian structured mesh is used
- Mesh is fixed
- Arbitrary body motion and deformation is allowed
- Efficient linear algebraic solvers (MultiGrid) can be used
- Problems with accurate governing equations approximation and boundary conditions satisfaction on cut-cells

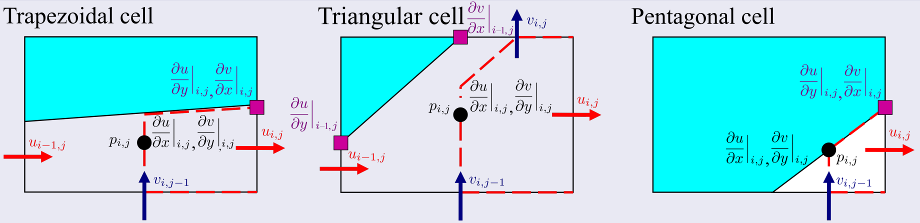
Level-Set STAGgered mesh method

- LS-STAG method is fully conservative IB method.
- 3 or 4 staggered meshes (in 2D case).

Cheny Y., Botella O. The LS-STAG method for the computation of incompressible viscous flows in complex moving geometries with good conservation properties. J. Comput. Phys. 229, 1043–1076 (2010)

LS-STAG cut-cells in 2D

3 elementary types of cut-cells


Trapezoidal cell

Triangular cell

Pentagonal cell

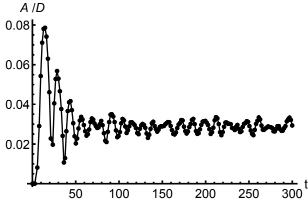
Navier — Stokes (NS) equations

- Mass, momentum and kinetic energy preservation.
- Unified discretization in all cells.
- 5-point structure of the stencil.
- Conditions on the Γ^{ib} are embedded in the discretization.

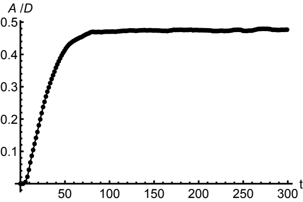
Development of LS-STAG

- Non-newtonian fluids (Cheny & Botella).
- Turbulent flows simulation: RANS, LES, DES approaches for Spalart — Allmaras, $k-\varepsilon$, $k-\omega$, $k-\omega SST$ models (Puzikova)
- 3D case (...)

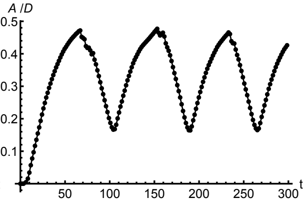
Amplitude (OpenFOAM, Vortex Method), $\mathrm{Re} = 1000$



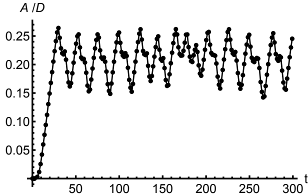
$\mathrm{St} = 0.190$



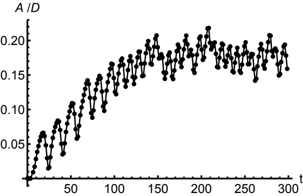
$\mathrm{St} = 0.205$



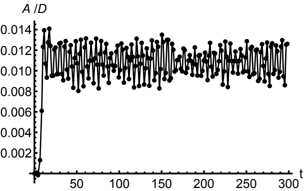
$\mathrm{St} = 0.213$



$\mathrm{St} = 0.230$



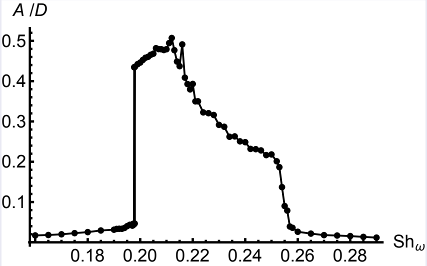
$\mathrm{St} = 0.250$



$\mathrm{St} = 0.285$

Hysteresis simulation $\mathrm{Re} = 1000$

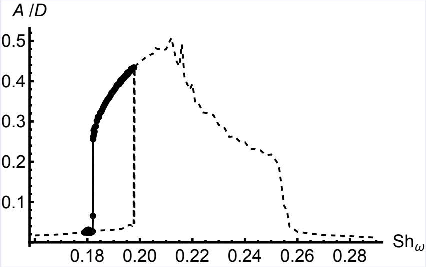
'Direct' simulation



$\mathrm{St} = 0.160\ldots0.290$ (max $\frac{A}{D} \approx 0.52$)

Sharp increase of the amplitude at $\mathrm{St} \approx 0.198$

Hysteresis simulation



$t = 0\ldots50: \quad \mathrm{St} = 0.210$ ($\frac{A}{D} \approx 0.47$)

$t = 50\ldots200: \ \mathrm{St} = 0.178\ldots0.198$

Dotted line — 'direct' simulation

1. Klamo, J.T., Leonard, A. and Roshko, A. The effects of damping on the amplitude and frequency response of a freely vibrating cylinder in cross-flow. *J. of Fluids and Struct.* (2006) **22**: 845–856.
2. Klamo, J.T., Leonard, A. and Roshko, A. On the maximum amplitude for a freely vibrating cylinder in cross flow. *J. of Fluids and Struct.* (2005) **21**: 429–434.

Flow around fixed cylinder

Results of flow simulation around circular cylinder, $\mathrm{Re} = 150$

	C_x	St	C_y^{ampl}
Experiment	$1.15 \ldots 1.45$	$0.175 \ldots 0.195$	$0.50 \ldots 0.65$
OpenFOAM	1.44	0.190	0.60
Kratos	1.20	0.190	0.53
Vortex Method	1.31	0.174	0.48
LS-STAG	1.32	0.185	0.55

Flow simulation around oscillating cylinder

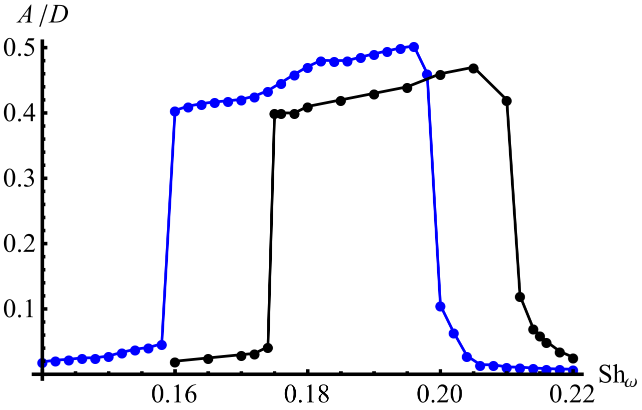
Problems with Kratos

- error in pressure calculation
- high C_y^{ampl} leads to instability

Problems with LS-STAG

- high computational cost
- no parallel version (yet)

Amplitude of oscillations (Wind resonance)



Amplitude of oscillations of circular cylinder, $\mathrm{Re} = 150$

▬ OpenFOAM ▬ Vortex Method,

Time of Computations

Physical time: 200
Processors: Intel Core i7
2.4 GHz

Regime	Kratos	LS-STAG
No resonance	55/16	180
Max amplitude	—	250

Computational time (in hours) for OpenFOAM

Regime	1 CPU	2 CPU	4 CPU	8 CPU	16 CPU
No resonance	58.1	36.0	24.3	15.5	9.8
Max amplitude	74.4	45.8	30.1	19.6	13.7

Computational time (in hours) for vortex element method

Regime	1 CPU	2 CPU	4 CPU	8 CPU	16 CPU
No resonance	41.3	22.6	12.0	7.1	4.7
Max amplitude	63.4	34.7	17.9	10.1	6.6

Comparison of different methods

	OpenFOAM	Vortex Method	Kratos	LS-STAG
Time	±	+	+	−
Accuracy	+	±	−	+
Body motion	±	+	±	+
Parallelization	+	+	−*	−*
Auto time step	+	−	−	±
Turbulent flows	+	−	−	+
3D case	+	−*	+	−*

* — this property is available for some special cases or ‘under construction’

Conclusion

- OpenFOAM usage is preferable for numerical simulation for wide range of hydrodynamic problems (including FSI-problems), when we need to investigate the behavior of the system in wide range of parameters.
- Other methods and codes can be more efficient in comparison with OpenFOAM for solving of some particular problems.