University of Sheffield

Immersed Boundary Method

Cell-Fluid Interactions

Matthew Leach 12/09/2023

Session Overview

1. Presentation describing the Immersed Boundary Method (~20 mins)
    a. What problem does it solve?
    b. How does it work?
    c. How do I use it?

2. Practical session (~2 hours)
    a. Simple immersed boundary simulations
    b. Multi-cellular simulations
    c. Introducing fluid sources

Immersed Boundary Method: The why

The Basics

The immersed boundary method is a technique for simulating fluid-structure interactions. The **boundaries** are the edges/outer surfaces of the structure. The structure is **immersed** in the fluid.

The method is quite general, the *structure* could be a heart valve, an aeroplane or in our case, a cell. The *fluid* could be air, water, serum, or any number of other things.

Challenges

It is not trivial to simulate fluid-structure interactions because of the *two-way coupling* of the system. The flow of **the fluid exerts a force on the structure**, but **the structure also influences the fluid**.

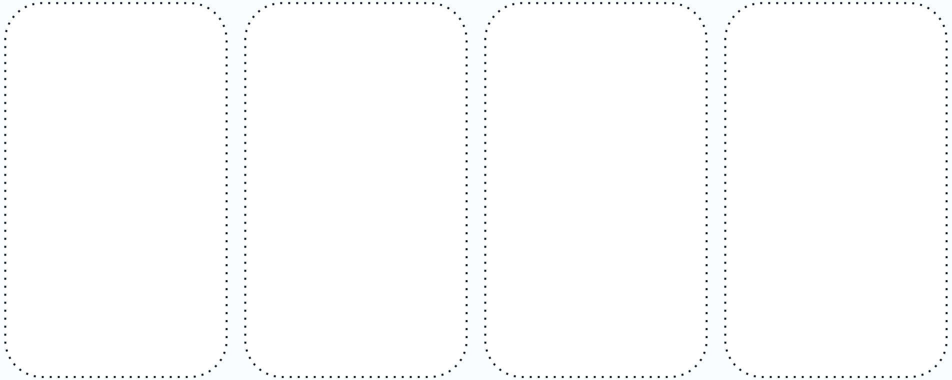
Example Applications

- Modelling biofilm processes *(Dillon, R., Fauci, L., Fogelson, A. and Gaver III, D., 1996. Modeling biofilm processes using the immersed boundary method. Journal of Computational Physics, 129(1), pp.57-73.)*
- Hemodynamics *(Ames, J., Puleri, D.F., Balogh, P., Gounley, J., Draeger, E.W. and Randles, A., 2020. Multi-GPU immersed boundary method hemodynamics simulations. Journal of computational science, 44, p.101153.)*
- Modelling cells interacting with structural environment features e.g. micro-channels *(Leong, F.Y., Li, Q., Lim, C.T. and Chiam, K.H., 2011. Modeling cell entry into a micro-channel. Biomechanics and modeling in mechanobiology, 10, pp.755-766.)*

Immersed Boundary Method: The how

Modelling Cells - The Structure

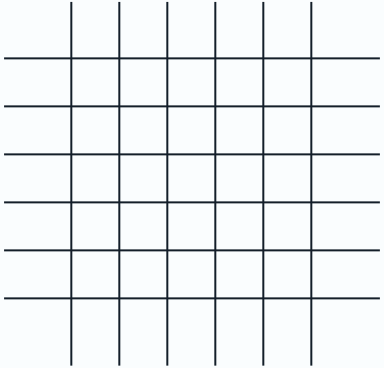
Cells in the immersed boundary method are essentially represented in the same way as in the vertex models you have already used - the cell membrane is described by a series of connected nodes/points. The nodes do not have to form a fully connected mesh.



Modelling ? - The Fluid

The fluid is simulated across a grid (known as Eulerian approach). The Navier-Stokes equations are solved at each of the grid points.

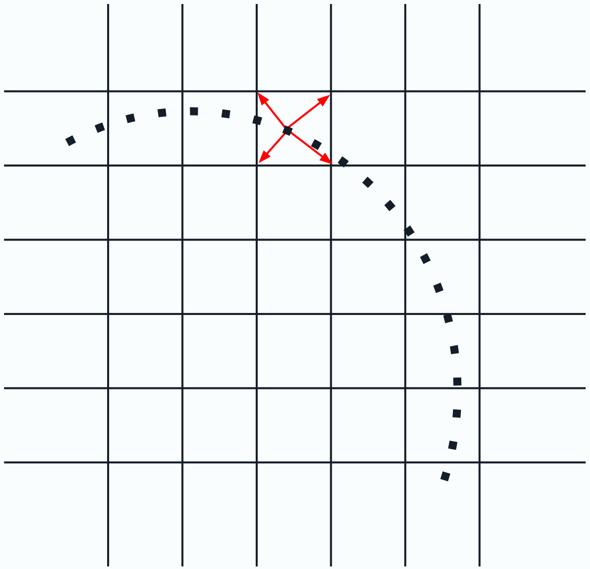
$$\rho\frac{\delta\mathbf{u}}{\delta x} = -\nabla p + \mu\nabla^2\mathbf{u} + \rho\mathbf{g}$$



University of Sheffield

Fluid-Structure Interactions

Forces are interpolated between nodes and the fluid grid.



Fluid Sources

- Fluid sources represent regions where fluid is entering the simulation domain, e.g. if simulating a segment of a blood vessel, one end would be a source as there is a positive flux.
- In simple terms, fluid will flow away from a source point.

The Immersed Boundary Method: Chaste

Features

- Two new mesh generators: *superellipse* and *palisade*
- Fluid simulation handled behind the scenes
- Adjustable fluid grid resolution
- Point-like fluid sources
- Optional normally distributed noise across the field
- Originally developed by Fergus

Will be merged into the main code base very soon!

Anatomy of an Immersed Boundary Sim

1. Generate a mesh
2. Set the fluid grid resolution
3. Generate a cell population
4. Add fluid sources
5. Add an ImmersedBoundarySimulationModifier
6. Add intra-cellular and inter-cellular force models
7. Go!
8. Visualise results with paraview

Practical Session Format

Session Overview

1. **Getting set up**
    a. Required software - docker, paraview
    b. Checking out the immersed boundary code
2. **Simple simulations**
    a. Generating an immersed boundary element
    b. Adjusting membrane properties
3. **Multi-cellular Simulations**
    a. Adding more cells
    b. Laminas
    c. Cell variations
    d. Intercellular forces
4. **Fluid Sources**
    a. Introducing a fluid source
    b. Experimenting with fluid source parameters
    c. Explore the relationship between fluid sources & membrane stiffness

Any questions?

m.leach@sheffield.ac.uk