We present a simple approach to simulate interaction between fluid and thin shell. Instead of solely considering one way coupling, we take both forces from fluid to thin shell and from thin shell to fluid into account, which is regarded as strong coupling. Such problem is also discussed in Carlson and Fedkiw. We borrow idea from Immersed Boundary Method (IBM) to solve this problem.

We applied the following process per time step for solving Navier-Stokes equation numerically: add force, mass conservation, advection.

\[
\frac{\partial u}{\partial t} + (u \cdot \nabla) u = -\frac{1}{\rho} \nabla p + \nu \nabla^2 u + f
\]

\[\nabla \cdot u = 0\]

Instead of solving the Poisson equation in spatial domain, we do it in frequency domain and applied periodic boundary condition. We demonstrate a simple situation in two domains below.

By using IBM, we add two more steps to the original fluid solver: add strain force to the fluid field and advect thin shell in the velocity field. 

- **Adding strain force**
  We model thin shell as set of Lagrangian points and compute internal stress on those points. In order to apply it as external force to the fluid field, we introduce Dirac delta function as filter model. The filtering process can be described as the following function:

\[
F = \int_{\Gamma} \delta(x - X(s,t)) ds
\]

\[
\delta(x) = \begin{cases} 
\frac{1}{2h} (1 + \cos(\frac{\pi x}{2h})), & x \leq 2h \\
0, & x > 2h
\end{cases}
\]

- **Advecting thin shell**
  We use two different scheme for fluid property and thin shell points in advection step. For fluid property, we apply Semi-Lagrangian approach for stability. Explicit RK 4 is utilized for thin shell points in order to pursue more accuracy. Thin shell may suffer overstretching sometimes if parameters are not tuned properly. However, one can overcome this by applying strain constraint directly or using more complicated nonlinear material model.

We have implemented IBM in 2D and several fast Poisson solvers including iterative solver Geometry Multi Grid, Conjugate Gradient and direct solver FFT. For more information, please refer to our online short paper. The link is http://www-scf.usc.edu/~yufengzh/doc/ibm.pdf

**Reference**