The Accurate and Efficient Numerical Simulation of General Fluid-Structure Interaction ——A Unified Finite Element Method

Yongxing Wang

2nd year PhD student in University of Leeds
Supervisors: Peter Jimack, Mark Walkley

scywa@leeds.ac.uk

June 5, 2016
Outline

1. Strategies of Finite Element Methods for FSI
2. A new Unified Finite Element Method (UFEM)
3. Numerical Examples
4. Conclusion and Future Work
Strategies of Finite Element Methods for FSI

- Strategies of Finite Element Methods for FSI
- A new Unified Finite Element Method (UFEM)
- Numerical Examples
- Conclusion and Future Work
How to handle the mesh: fitted and unfitted

**Figure:** Fitted Mesh

- Fitted boundary
- Geometry of the boundary is captured
- Nodes on the boundary are shared

Yongxing Wang (University of Leeds)
UFEM for FSI
June 5, 2016 4 / 39
How to handle the mesh: fitted and unfitted

Figure: Unfitted Mesh

- Unfitted boundary
- Solid and fluid have separate meshes
How to couple the interaction between fluid and solid

Partitioned/Segregated Methods
The fluid and solid equations are solved sequentially...

Monolithical/Fully-Coupled Methods
The fluid and solid equations are solved simultaneously...
How to couple the interaction between fluid and solid

Partitioned/Segregated Methods

The fluid and solid equations are solved sequentially...

Monolithical/Fully-Coupled Methods

The fluid and solid equations are solved simultaneously...

- Partitioned/Segregated Methods using Fitted meshes
- Partitioned/Segregated Methods using Unfitted meshes
- Monolithical/Fully-Coupled Methods using Fitted meshes
- Monolithical/Fully-Coupled Methods using Unfitted meshes
A general FEM scheme for FSI that can

- Cheaply simulate large deformation
- Simulate from very soft to very hard solid in one scheme (for example: $E = 10^0 \sim 10^{12} Pa$)
- Efficiently, accurately, robustly do 3D simulation.
A general FEM scheme for FSI that can
- Cheaply simulate large deformation
- Simulate from very soft to very hard solid in one scheme (for example: $E = 10^0 \sim 10^{12} Pa$)
- Efficiently, accurately, robustly do 3D simulation.

mesh:
- fitted, ALE, moving mesh, re-mesh. **unfitted two meshes**

coupling strategy:
- partitioned, IBM, IFEM, existing monolithic. **kind of monolithic**
Monolithical/Fully-Coupled Methods Using Unfitted Meshes

\[
\begin{bmatrix}
K^f & B \\
B^T & 0
\end{bmatrix}
\begin{bmatrix}
v^f \\
p
\end{bmatrix} =
\begin{bmatrix}
r^f \\
0
\end{bmatrix}
\]

\[K^s v^s = r^s\]

s.t. \[Dv^f = v^s\]

Figure: illustration by a discretized system
The above equations can be expressed as:

\[ Ax = b, \quad s.t. \quad Dv^f = v^s. \]

where

\[
A = \begin{bmatrix} K^f & B & 0 \\ B^T & 0 & 0 \\ 0 & 0 & K^s \end{bmatrix}, \quad x = \begin{pmatrix} v^f \\ p \\ v^s \end{pmatrix} \quad \text{and} \quad b = \begin{pmatrix} r^f \\ 0 \\ r^s \end{pmatrix}.
\]

Or using Lagrangian Multiplier

\[
\text{minimize} \quad E(x, \lambda) = \frac{1}{2}x^T Ax - x^T b + (Dv^f - v^s)^T \lambda
\]

The last term can also be expressed as:

\[
(Dv^f - v^s)^T \lambda = x^T \begin{pmatrix} D^T \lambda \\ 0 \\ -\lambda \end{pmatrix}
\]
Taking derivatives of $E(x, \lambda)$ gives the equations for the whole FSI system.

\[
\begin{bmatrix}
K^f & B & 0 & D^T \\
B^T & 0 & 0 & 0 \\
0 & 0 & K^s & -I \\
D^T & 0 & -I & 0
\end{bmatrix}
\begin{bmatrix}
v^f \\
p \\
v^s \\
\lambda
\end{bmatrix}
=
\begin{bmatrix}
r^f \\
0 \\
r^s \\
0
\end{bmatrix}
\]
Monolithical/Fully-Coupled Methods Using Unfitted Meshes

Taking derivatives of $E(x, \lambda)$ gives the equations for the whole FSI system.

$$
\begin{bmatrix}
K^f & B & 0 & D^T \\
B^T & 0 & 0 & 0 \\
0 & 0 & K^s & -I \\
D^T & 0 & -I & 0
\end{bmatrix}
\begin{bmatrix}
v^f \\
p \\
v^s \\
\lambda
\end{bmatrix}
= 
\begin{bmatrix}
r^f \\
0 \\
r^s \\
0
\end{bmatrix}
$$

- Solid equations are expressed as displacement...
- Continuous equations, weak forms (mortar integral)...

Frank P. T. Baaijens (2001)
A fictitious domain/mortar element method for fluid-structure interaction

A mortar approach for Fluid-Structure interaction problems: Immersed strategies for deformable and rigid bodies

A fictitious domain/distributed Lagrange multiplier based fluid-structure interaction scheme with hierarchical B-Spline grids
A new Unified Finite Element Method (UFEM)

- Strategies of Finite Element Methods for FSI
- A new Unified Finite Element Method (UFEM)
- Numerical Examples
- Conclusion and Future Work
The following equations are solved for the whole FSI system:

\[
\begin{bmatrix}
K_f & B \\
B^T & 0
\end{bmatrix}
\begin{bmatrix}
v_f \\
p
\end{bmatrix} =
\begin{bmatrix}
r_f \\
0
\end{bmatrix}
\]

\[K^{s-f}v^s = r^{s-f}\]

s.t. \[Dv^f = v^s\]
The following equations are solved for the whole FSI system:

\[
\begin{bmatrix}
K^f & B \\
B^T & 0
\end{bmatrix}
\begin{bmatrix}
v^f \\
p
\end{bmatrix}
= 
\begin{bmatrix}
r^f \\
0
\end{bmatrix}
\]

\[K^{s-f} v^s = r^{s-f}\]

s.t. \[Dv^f = v^s\]
The Weak Form of UFEM

Weak form:

\[ \rho_f \left( \frac{dv}{dt}, \delta v \right)_\Omega + \left( \tau^f, \nabla \delta v \right)_\Omega - (p, \nabla \cdot \delta v)_\Omega - (\nabla \cdot v, \delta p)_\Omega \\
+ \left( \rho^s - \rho^f \right) \left( \frac{dv^s}{dt}, \delta v \right)_\Omega^s + \left( \tau^s - \tau^f, \nabla \delta v \right)_\Omega^s \\
= \left( \tau^f \cdot n, \delta v \right)_{\partial \Omega} + \rho^f (g, \delta v)_\Omega + \left( \rho^s - \rho^f \right) (g, \delta v)_\Omega^s \\
\]

where \( \mathbf{g} \) is acceleration due to gravity.
The Weak Form of UFEM

Weak form:

\[
\rho^f \left( \frac{dv}{dt}, \delta v \right)_{\Omega} + \left( \tau^f, \nabla \delta v \right)_{\Omega} - (p, \nabla \cdot \delta v)_{\Omega} - (\nabla \cdot v, \delta p)_{\Omega} \\
+ (\rho^s - \rho^f) \left( \frac{dv^s}{dt}, \delta v \right)_{\Omega^s} + \left( \tau^s - \tau^f, \nabla \delta v \right)_{\Omega^s} = \left( \tau^f \cdot n, \delta v \right)_{\partial \Omega} + \rho^f (g, \delta v)_{\Omega} + (\rho^s - \rho^f) (g, \delta v)_{\Omega^s}
\]

where \( g \) is acceleration due to gravity.

which comes from the strong form on the following page...
Control Equations for the FSI System

**Fluid equations in $\Omega^f$**

\[
\rho^f \frac{d\mathbf{v}^f}{dt} - \nabla \cdot \mathbf{\sigma}^f = \rho^f \mathbf{g}
\]  
(1)

\[
\nabla \cdot \mathbf{v}^f = 0
\]  
(2)

\[
\mathbf{\sigma}^f = \mu^f \left( \nabla \mathbf{v}^f + (\nabla \mathbf{v}^f)^T \right) - p^f \mathbf{I}
\]  
(3)

**Solid equations in $\Omega^s$**

\[
\rho^s \frac{d\mathbf{v}^s}{dt} - \nabla \cdot \mathbf{\sigma}^s = \rho^s \mathbf{g}
\]  
(4)

\[
det \mathbf{F} = 1
\]  
(5)

\[
\mathbf{\sigma}^s = \mu^s \left( \mathbf{F} \mathbf{F}^T - \mathbf{I} \right) - p^s \mathbf{I}
\]  
(6)
Fluid equations in $\Omega^f$

$$\rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) - \nabla \cdot \sigma = \rho \mathbf{g}$$

$$\nabla \cdot \mathbf{v} = 0$$

Splitting scheme

Step 1:

$$\frac{\mathbf{v}^* - \mathbf{v}^n}{\Delta t} + \mathbf{v}^* \cdot \nabla \mathbf{v}^* = 0$$

Step 2:

$$\rho \frac{\mathbf{v}^{n+1} - \mathbf{v}^*}{\Delta t} - \nabla \cdot \sigma^{n+1} = \rho \mathbf{g}$$

$$\nabla \cdot \mathbf{v}^{n+1} = 0$$
Adaptive mesh with hanging nodes

In order to improve the computational efficiency, an adaptive spatial mesh with hanging nodes is used in all the numerical experiments.

Figure: Configuration of leaflet and contours of pressure on the adaptive fluid mesh
Numerical Examples

- Strategies of Finite Element Methods for FSI
- A new Unified Finite Element Method (UFEM)
- Numerical Examples
- Conclusion and Future Work
Ex1. Oscillation of a flexible leaflet in a channel

Figure: Computational domain and boundary conditions

\[ U = 15y(2 - y)\sin(2\pi t) \]
Ex1. Oscillation of a flexible leaflet in a channel

Table: Fluid and leaflet properties and domain size

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Leaflet</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L = 4.0m$</td>
<td>$w = 0.0212m$</td>
</tr>
<tr>
<td>$H = 1.0m$</td>
<td>$h = 0.8m$</td>
</tr>
<tr>
<td>$\rho_f = 100,kg/m^3$</td>
<td>$\rho_s = 100,kg/m^3$</td>
</tr>
<tr>
<td>$\mu_f = 10,N/m^2s$</td>
<td>$\mu_s = 10^7,N/m^2s$</td>
</tr>
</tbody>
</table>

1. Influence of the solid mesh size (fixed fluid mesh size).
2. Time convergence (medium meshes, ratio of mesh sizes $\approx 1.5$).
3. Mesh convergence (ratio of mesh sizes $\approx 1.5$).
4. Comparison with the results of monolithical methods.
5. Tests of a range of parameters from very soft to very hard solid.
Ex1. Oscillation of a flexible leaflet in a channel
Comparison with the results of monolithical methods

Figure: Evolution of horizontal and vertical displacement at top right corner of the leaflet
(Good agreement with the results using monolithical methods)
Tests of a range of parameters

Based on:

\[ \bar{\mu}^s = 10^3 \]
\[ \rho^r = 1 \]
\[ Re = 100 \]
\[ Fr = 0 \]

\[ \bar{\mu}^s = \frac{\mu^s}{\rho^f U^2} \]
\[ \rho^r = \frac{\rho^s}{\rho^f} \]
\[ Re = \frac{\rho^f U H}{\mu^f} \]
\[ Fr = \frac{g H}{U^2} \]
Tests of a range of parameters

\[ \rho^r = 10 \]
\[ \rho^r = 5 \]
\[ \rho^r = 3 \]
\[ \rho^r = 1 \]
\[ t/T = 1.0 \]

Red: Fr=0

Green: Fr=0.1

\[ \rho^r = 3 \]
\[ \rho^r = 5 \]
\[ \rho^r = 10 \]
\[ t/T = 1.2 \]
Ex2. Oscillation of a flexible leaflet across the flow direction

\[ \mu^f = 1.82 \times 10^{-4}, \rho^f = 1.18 \times 10^{-3}, \mu^s = 9.2593 \times 10^{-5} \text{ and } \rho^s = 0.1. \]
Ex2. Oscillation of a flexible leaflet across the flow direction
Ex2. Oscillation of a flexible leaflet across the flow direction

Figure: Contours of leaflet displacement and fluid pressure at $t=3.63$
Ex2. Oscillation of a flexible leaflet across the flow direction

**Figure:** Contours of leaflet displacement and fluid pressure at $t=5.44$
Displacement of leaflet tip as a function of time

<table>
<thead>
<tr>
<th>Author</th>
<th>Magnitude</th>
<th>Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall [1999]</td>
<td>1.12 ～ 1.32</td>
<td>2.78 ～ 3.22</td>
</tr>
<tr>
<td>Dettmer and Perić [2004]</td>
<td>1.1 ～ 1.4</td>
<td>2.96 ～ 3.31</td>
</tr>
<tr>
<td>C.Kadapa [2015]</td>
<td>1.26</td>
<td>3.22</td>
</tr>
<tr>
<td>Present</td>
<td>1.356</td>
<td>2.94</td>
</tr>
</tbody>
</table>

![Graph showing displacement of leaflet tip as a function of time.](image-url)
Ex3. Solid disc in a cavity flow

\[ \bar{u} = 1.0 \]

\( \bar{\mu}^s = 10^2, \rho^r = 1, Re = 100, Fr = 0. \)
Solid disc in a cavity flow
Comparison between UFEM and IFEM

Figure: Velocity norm using UFEM (left) and IFEM (right)

(a) t=5.0 s

(b) t=25 s

Figure: Velocity norm using UFEM (left) and IFEM (right)
A general FEM scheme for FSI that can

- Cheaply simulate large deformation
- Simulate from very soft to very hard solid in one scheme (for example: $E = 10^0 \sim 10^{12} Pa$)
- Efficiently, accurately, robustly do 3D simulation.
A general FEM scheme for FSI that can
- Cheaply simulate large deformation
- Simulate from very soft to very hard solid in one scheme (for example: $E = 10^0 \sim 10^{12}\ Pa$)
- Efficiently, accurately, robustly do 3D simulation.

Almost, based on current numerical scheme and 2D results. Halfway through it, validation in 3D.
The proposed UFEM is a method for general fluid-structure interaction (Newtonian and Oldroyd-B fluid, linear elastic and incompressible Neo-Hooken model have been implemented), which unifies the fluid and solid equations and can model a range of solid materials from very soft to very hard.

The UFEM only solves for velocity and pressure as unknowns, which is cheaper than monolithical methods, but does not lose stability and accuracy which can be seen from the numerical experiments.
Future Work

- Implementation in 3D using adaptive mesh with hanging nodes.
- Preconditioned efficient solver for the UFEM algebraic system.
- Second order splitting scheme in time.
- A $\theta$-scheme to split the terms of solid intergral and arrange them on both sides of the equation.
Thank You !
Partitioned/Segregated Methods Using Fitted Meshes

\[ \Omega = \Omega^f \cup \Omega^s \]

Solve fluid in \( \Omega^f \)

Solve solid in \( \Omega^s \)
Immersed FEM (IFEM): Not solve solid equations (classical IFEM). The solid equations are arranged on the right-hand side of fluid equations as an FSI force, and fluid equations on the augmented domain $\Omega = \Omega^f \cup \Omega^s$.

- Fluid domain $\Omega^f$?
- Boundary condition (BC)?
- Solve fluid equations in $\Omega$?
- BC for solid problem?
Partitioned/Segregated Methods Using Unfitted Meshes

Immersed FEM (IFEM): Not solve solid equations (classical IFEM). The solid equations are arranged on the right-hand side of fluid equations as an FSI force, and fluid equations on the augmented domain $\Omega$ are solved.
An efficient solver for the fully coupled solution of large-displacement fluid-structure interaction problems

Solvers for large-displacement fluid-structure interaction problems: segregated versus monolithic approaches

U. Küttler, M. Gee, Ch. Förster, A. Comerford and W. A. Wall (2010)
Coupling strategies for biomedical fluid-structure interaction problems

Richard L. Muddle, Milan Mihajlović, Matthias Heil (2012)
An efficient preconditioner for monolithically-coupled large-displacement fluid-structure interaction problems with pseudo-solid mesh updates