TECHNICAL NOTE:
IMMERSED BOUNDARY METHOD (BODY FORCE) FOR FLOW AROUND THIN BODIES WITH SHARP EDGES

T. M. Y. S. Tuanya1, S. Takeuchi3, T. Kajishima2 and A. Ueyama2

1 Department of Engineering Design and Manufacture, Faculty of Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia
2 Department of Mechanical Engineering, Osaka University, 2-1 Yamada-oka, Suita City, Osaka 565-0871, Japan
3 Department of Mechanical Engineering, Tokyo University, 7-3-1 Hongo, Bunkyo-ku, Tokyo, 113-8656, Japan
E-mail: tyusoff@um.edu.my

ABSTRACT

Efficient methods to identify immersed boundary nodes and handling of thin bodies with sharp edges are introduced as an improved immersed boundary method. Particular attention is focused on the treatment of interaction with thin structure and near the trailing edge. It is applied to 2-D flow fields involving fixed rigid object. The immersed boundary method uses a body force proportional to a solid volume fraction to couple the motions of solid and fluid at the interface. The improved method is proved to be applicable to various arbitrary objects including those with thin profile and sharp edges. The present method is applied to a uniform flow field around a rigid 2-D thin airfoil. The computational results had shown the capability of the method to solve the complexity faced by this geometry and also other arbitrary shaped objects.

Key words: Immersed Boundary Method, Fluid-Solid interaction, Computational Fluid Dynamics, Thin profile, Sharp edges.

1. INTRODUCTION

Various types of flow with environmental and biological background involve flow with complex and thin bodies. Examples of the flow include leaves falling through air, flapping of the flag in the wind and flow past micro organisms. Traditionally, simulation of fluid-structure interactions are solved using body-fitted methods based on arbitrary Lagrangian-Eulerian (ALE) approach. Flow can be resolved along the object surface giving high resolution velocity and pressure field near the boundary layer. However for objects with complex geometry, the mesh generation would not be a straightforward affair, and the need to re-mesh the entire computational domain each time the object displaced or deformed increases the computation load significantly. On the other hand, solving the governing equations on a fixed Cartesian grid for flow around an arbitrary and thin object has been attracting interest due to relatively simple approach in computation technique and low computing cost. In this approach, flow with an arbitrarily shaped object is solved by a rectangular grid and the interface of the two phases is solved through a special procedure taking consideration of the occupied cells. This approach is generally known as immersed boundary method (IBM). IBM has been applied to various biological and medical field simulations such as blood flow through the heart. IBM was first introduced by Peskin (1972), where the effect of a stationary or moving boundary is accounted for by introducing an external force field in the equations of motion which ensure that the fluid satisfies the no-slip conditions on solid boundaries. Deformable thin object such as filament in flowing film also has been incorporated into IBM (Zhu and Peskin 2003; 2007). For the interaction between the filament and the soap film, a smoothed approximation to the Dirac delta function is used. This delta function approximation is used to interpolate fluid velocity and also apply the force to the fluid. Kajishima et al. (2001) have developed independently an immersed boundary method with body force type. The force between the solid and fluid is derived efficiently by a volume function of the solid volumetric fraction and the relative velocities of the two phases. The authors have used Kajishima’s method to simulate thin body due to efficient treatment at the boundary. However careful treatment for momentum exchange is needed for an arbitrarily shaped and thin object.

The present study intends to show a new series of additions in fluid-structure interaction utilizing IBM (body force) for thin bodies. By simply specifying one surface as external and another surface as internal, immersed boundary method (body force) can be utilized to simulate flows around thin body. A surface digitizer for arbitrary shape is also essential for simulating non-spherical complex object efficiently. The method is then applied to 2-D flow fields with thin body to evaluate the applicability of the method.

2. GOVERNING EQUATION AND NUMERICAL METHODS

The governing equations for incompressible fluid flow are the continuity equation and Navier-Stokes equations:
\[ \nabla \cdot u_f = 0 \]  
\[ \frac{\partial u_f}{\partial t} + u_f \cdot \nabla u_f = -\frac{1}{\rho_f} \nabla p + \nu \nabla \left( \nabla u_f + \left( \nabla u_f \right)^T \right) \]  
(1)
(2)

where \( u_f \) is fluid velocity, \( p \) is pressure, \( \rho_f \) density and \( \nu \) kinematic viscosity. Both density and kinematic viscosity are kept constant for this study. Equations (1) and (2) then solved using finite difference method with a second order accuracy in space and time. For cells which partially occupied by the solid structure, Kajishima et al (2001) proposed an immersed boundary method that solves the momentum exchange at the fluid-structure boundary. This method is briefly described as follows.

A velocity field \( u \) is introduced by volume averaging the local fluid velocity \( u_f \) and the local solid phase velocity \( \nu \) in each cell.

\[ u = (1 - \alpha)u_f + \alpha \nu \]  
(3)

where \( 0 \leq \alpha \leq 1 \) is the local solid volumetric fraction in a cell. The following hyperbolic-tangent function is used to evaluate the volume fraction:

\[ \alpha = \frac{1}{2} \left[ 1 - \tanh \left( \frac{\delta n}{\sigma \lambda \Delta} \right) \right] \]  
(4)

\[ \lambda = |n_x| + |n_y| + |n_z| \]  
(5)

\[ \sigma = 0.05 \left( 1 - \lambda^2 \right) + 0.3 \]  
(6)

where \( n = (n_x, n_y, n_z) \) is a normal outward unit vector at a surface element and \( \delta n \) is a signed distance from the cell centre to the surface element. This solution is called a surface digitizer developed (Yuki et al, 2007). The fluid-structure interaction then can be solved at the interface with \( u \) by assuming that the velocity field \( u \) follows the Navier-Stokes equations:

\[ \frac{\partial u}{\partial t} = -\nabla p + H_u \]  
(7)

\[ H_u = -u \cdot \nabla u \]  
(8)

a time advancement scheme for \( u \) can be proposed as follow,

\[ u_n = u^n + \Delta t \left( \frac{3}{2} H_u^n - \frac{1}{2} H_u^{n-1} \right) \]  
(9)

\[ \nabla^2 p^{n+1} = \frac{\nabla \cdot u_f}{\Delta t} \]  
(10)

\[ u^{n+1} = u^n - \Delta t \nabla p^{n+1} \]  
(11)

where superscripts \( n \) represent time, \( \Delta t \) the time increment and \( u_i \) is scalar velocity. Time advancement is the carried by second-order Adam-Bashforth method and FSM for a single-phase fluid. A Poisson equation is then solved with the divergence of the scalar velocity \( u_i \) as a source term to provide a pressure \( p^{n+1} \), which is then used to correct the scalar velocity, providing a divergence free velocity \( u^{n+1} \) and integration proceed to the next time step.

3. IBM FOR THIN PROFILE OBJECT

To apply IBM (body force) on thin profile object, new method is developed to identify surrounding fluid cells nearest to Lagrangian point on the object’s surface. For each \( n^{th} \) node on the surface of the object, the method selects a \( 3 \times 3 \)-mesh around the cell to which the node belongs. The same procedure is repeated for all nodes and redundant cells will be removed. Local solid volumetric fraction of each cells are calculated using equations (4-6). Since the thin object has 2-sides, 1 side must be treated as external surface and another side as internal surface. Figure 1 shows the selected fluid cells near the thin body.

![Fig. 1: Fluid cells selected for calculation of local volumetric fraction](image)

4. SIMULATION SETUP

Two simulation cases have been investigated. Case 1 is simulating a flow through a crescent shaped thin body and Case 2 with thin airfoil. Both bodies are initially placed in a fluid at rest and the thin airfoil with an attack angle of 15º. The fluid is subjected with an instantaneous start at the inlet with velocity \( U_0 \). Figure 2 and 3 show schematic of the computational domains for both cases and Table 1 and 2 shows the computational conditions. On top and bottom boundaries, a periodic boundary condition is used. At the inlet, a constant velocity is prescribed and at the outlet, gradient free condition is used.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Coarse mesh</th>
<th>Fine mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of grid point</td>
<td>200 × 150</td>
<td>400 × 300</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>Time increment, ( \Delta t )</td>
<td>0.005</td>
<td>0.0025</td>
</tr>
<tr>
<td>Grid size, ( \Delta x )</td>
<td>0.1</td>
<td>0.05</td>
</tr>
</tbody>
</table>
Table 2: Computational setup for thin airfoil (Case 2)

<table>
<thead>
<tr>
<th></th>
<th>Coarse mesh</th>
<th>Fine mesh</th>
<th>Extra fine mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of grid point</td>
<td>200 × 150</td>
<td>400 × 300</td>
<td>800 × 600</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>200</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>Time increment, Δt</td>
<td>0.005</td>
<td>0.0025</td>
<td>0.00125</td>
</tr>
<tr>
<td>Grid size, Δx</td>
<td>0.1</td>
<td>0.05</td>
<td>0.025</td>
</tr>
<tr>
<td>Angle of attack (α)</td>
<td>15º</td>
<td>15º</td>
<td>15º</td>
</tr>
</tbody>
</table>

Fig. 2: Schematic of computational domain for crescent shaped thin body

4. RESULTS AND DISCUSSION

4.1 Crescent shaped thin body

For crescent shaped thin body, 2 possible configurations were tested for the selection of further mesh refinement. First configuration is to assign surface facing the incoming flow as external surface (left external – LE) and second configuration assigns surface opposing the incoming flow as external surface (right external – RE). Figure 4 show the velocity vectors for both configurations. From the plots, we can observe the virtual thickness represented by the empty spaces. From there, we can see that the configuration of LE is much better in simulating the flow over thin body.

(a) Left surface acting as external  

(b) Right surface acting as external

Fig. 4: Velocity vectors plot for LE and RE on crescent shaped thin body

Fig. 5: Velocity vectors plot of refined mesh for crescent shaped thin body
The virtual thickness area is identified as an area that need to be improved to obtain reasonable solution. Further refinement of the mesh is done to see the effect on the virtual thickness. Figure 5 shows the velocity vectors for the refined mesh of crescent shaped thin body. The virtual thickness is reduced significantly. With further refinement, the virtual thickness will become negligible to the dimension of the thin body.

4.2 Thin airfoil

To study the effect of mesh refinement on the virtual thickness, thin airfoil is subjected under the same flow condition as crescent shaped thin body. The mesh were refined up to 3 times and the coefficient of lift is measured for all three instances.

![Velocity vectors for refined mesh of crescent shaped thin body.](image)

Figure 5: Velocity vectors for the refined mesh of crescent shaped thin body.

![Velocity vectors for refined mesh of thin airfoil.](image)

Figure 6: Velocity vectors for thin airfoil with 3 different mesh sizes.

![Velocity vectors for coarse, fine, and extra fine mesh.](image)

(a) Coarse mesh, $\Delta x = 0.1$

(b) Fine mesh, $\Delta x = 0.05$

(c) Extra fine mesh, $\Delta x = 0.025$

CONCLUSION

A simple IBM (body force) method to simulate flow with thin bodies has been developed. Mesh refinement helps reduce the effect of virtual thickness to the flow and bring applicability of this method. However, the computation cost has been increased many fold due to very fine mesh in the whole computation domain. To obtain the same benefit of mesh refinement and IBM to solve flow with thin body, further improvement is needed. One of the approach is to utilize local mesh refinement at the overlapped fluid cells. With this, the same benefit can be obtained without the high computation cost resulted from high density mesh in the computational domain.

REFERENCES

vertical plane channel due to vortex shedding, JSME Int. J. Ser. B, 44-4, pp. 526-535.