APPLICATION OF FLUID-STRUCTURE INTERACTION SOLVERS TO TALL BUILDINGS

Aishe Zhang\textsuperscript{1,2}, Ming Gu\textsuperscript{3}, Deqian Zheng\textsuperscript{4}, Cuilan Gao\textsuperscript{5}, Weirui Liu\textsuperscript{5}

\textsuperscript{1} Vice Professor, State Key Laboratory for Disaster Reduction in Civil Engineering, Tongji University, Shanghai 200092, China, acclan@263.net
\textsuperscript{2} School of Civil Engineering, Shandong Jianzhu University, Ji’nan 250101, China
\textsuperscript{3} Professor, State Key Laboratory for Disaster Reduction in Civil Engineering, Tongji University, Shanghai 200092, China, minggu@mail.tongji.edu.cn
\textsuperscript{4} Ph.D Candidate, State Key Laboratory for Disaster Reduction in Civil Engineering, Tongji University, Shanghai 200092, China, zhengdeqian388@tom.com
\textsuperscript{5} Graduate Student, School of Civil Engineering, Shandong Jianzhu University, Ji’nan 250101, China, clgao@sdjzu.edu.cn

ABSTRACT

A numerical simulation approach for fluid-structure coupling on tall buildings is put forward by coupling two codes developed for flow simulation and structural dynamics in this paper. The interaction is described as the structural deformation as response to wind forces, resulting in a modification of the fluid flow domain. The partitioned solution method for structure-wind interaction problems realized by an iterative frame algorithm for controlling the simulation process and transferring the grid based data is presented. Some important phenomena such as vortex shedding and resonance have been captured.

KEYWORDS: FLUID-STRUCTURE INTERACTION (FSI), TALL BUILDINGS, AEROELASTICITY, PARTITIONED SOLUTION ALGORITHM, ARBITRARY LAGRANGEAN – EULERIAN METHOD

Introduction

The interaction of air flow and elastic structures plays an important role in civil engineering. The problem was studied widely through wind tunnel experiments and sometimes by monitoring in the field. In addition, the increase of the speed of modern computers and the development of the multidisciplinary solution algorithms necessitate the use of numerical approaches. Although the fluid-structure interaction (FSI) problems remain many challenges for the scientific computing, some progress has been obtained in the solution of these problems in recent years. The great potential for the application of the FSI technique is attracting more and more attention.

The computation of fluid-structure interaction problems requires the simultaneous solution of the coupled fluid and structural equations of motion. The coupling of the fluid and the structure occurs at the fluid and structure interface where common dynamic and kinematic conditions are imposed. The governing equations from the two fields have to be integrated. The methods of simulation of fluid-structure interaction problems are divided basically into two groups called partitioned and monolithic schemes (Ohayon and Felippa 1998, Van Loon et al 2007). In the partitioned schemes the governing equations of the fluid and the structure are integrated in time alternately in an isolated way. In the monolithic schemes the two fields are considered as a single entity, allowing to integrate in time the two subproblems simultaneously. The coupling of the fluid and the structure occurs at the fluid and structure
interface where the dynamic and kinematic coupling conditions are imposed. In portioned schemes, each subproblem can be solved by efficient solution algorithms in an individual way. Partitioned schemes also support the use of existing and highly developed software and offer a modular solution approach.

In this paper, an algorithm to simulate fluid-structure interaction using a partitioned procedure (Piperno et al 1995, Gluck et al 2003) is presented. An arbitrary Lagrangean – Eulerian (ALE) description is adopted for the fluid domain, while for the structural domain an updated Lagrangean formulation is considered. The loose sequential coupling methods proposed by (Farhat 1995) is used in this approach. The large nonlinear equations for FSI problems are done by using Block Gause-Seidel iterative method (Matthies 2002). The characteristics of the tall building structure’s pressures and displacements owing to the wind-induced vibration were obtained.

Mathematical Formulation

Fluid domain

The equations solved for the fluid domain are the incompressible Renolds-averaged Navier-Stokes equations and the continuity equation:

\[ \nabla \cdot \mathbf{u} = 0 \]  \hspace{1cm} (1)

\[ \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \mathbf{u} \mathbf{u} = -\nabla p + \rho \mathbf{u} \mathbf{u} \]  \hspace{1cm} (2)

Here, \( p \) is pressure, \( \mathbf{u} \) is velocity vector with \( U \) in the streamwise direction \( x \) and \( V \) in the lateral direction \( y \).

Structural domain

The fluid force obtained by solving Navier-Stokes equations is coupled with the motion of the building. The dynamical equation of the building represented by a discretized lumped-mass system is expressed as the following.

\[ [M] \ddot{x} + [C] \dot{x} + [K] x = \{f(t)\} \]  \hspace{1cm} (3)

where \([M]\), \([C]\), and \([K]\) are the system mass, damping and stiffness matrices, respectively. \( \{x\} \) is the streamwise displacement of the building, \( \{f(t)\} \) is the wind loads acted on the building. For the system, the damping ratio is \( \xi = 0.05 \). The equation of motion can be solved with various methods. In this study, Newmark method is used to solve the coupled equations.

Boundary conditions

For the fluid domain in the system, there are two types of boundaries: (1) stationary boundaries. These include the peripheral boundaries, such as the inlet boundary and outlet boundary; (2) moving boundaries. These include the interface between the fluid domain and the cylinder, which is determined by the shape of the moving cylinder. Besides the ordinary boundary conditions for the stationary boundaries, the moving boundary conditions should be used for the moving fluid domain contacting with the cylinder. Moreover, the moving boundary of the fluid domain should be compatible with the cylinder, that is, both sides of the interface should have the same value of displacements and velocities at each time step. For the computational fluid dynamical analysis, the boundary conditions used in Fluent code are summarized in Table 1.

Table 1. Name of BCs in Fluent 6.2

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Inlet</th>
<th>Outlet</th>
<th>Up</th>
<th>Down</th>
<th>Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name of BCs</td>
<td>Velocity Inlet</td>
<td>Pressure Outflow</td>
<td>Symmetry</td>
<td>Symmetry</td>
<td>Moving &amp; No Slip</td>
</tr>
</tbody>
</table>
Numerical Methodology

The domain of the simulation is presented in Figure 1. The computational domain is 25B in the streamwise direction and 17B in the lateral direction. The front face of the building is at a distance of 6H from the entrance flow boundary. The time step chosen as 0.001s is small enough to capture the vertex shedding phenomenon. The planar grid, as shown in Figure 2, consists of 7500 quadrilateral meshes. The governing equations (Eqs.(1) ~ (3)) for FSI problems are both time and space dependent. The equations need to be discretized before solution methods can be applied. In the present work, numerical solutions of the governing system of the coupled nonlinear partial difference equations are obtained by using Fluent code and the structural dynamical analysis. To take advantage of Fluent code and to realize an effective coupling algorithm, a partitioned solution approach was chosen (Gluck et al 2003).

![Fig. 1 Geometry and boundary condition of the domain](image1)

![Fig. 2 Meshing near the building](image2)

The simulation is based on an iterative frame algorithm, integrating both field solution methods in an implicit time-stepping procedure shown in Figure 3. The outer loop describes the temporal discretization of FSI problem. Within each time step, outer iterations between CFD and CSD simulations are performed until the global convergence is reached. The load for the CSD simulation is performed from the load resultant of the CFD computation. The boundary geometry is corrected according to the structural displacements computed by the CSD simulation. A segregated iterative solution scheme is implemented that is briefly summarized as follows.
CFD simulation: Initial conditions and boundary conditions are imposed to the fluid field. The fluid velocity and pressure fields are solved for the segregated scheme. The force vectors at the building surfaces are calculated.

Date transfer: The forces are transferred from CFD to CSD by using a conservative interpolation method (Farhat et al 1998).

CSD simulation: The fluid forces are applied and the initial and boundary conditions for the equation of motion of the building are imposed. The displacements at the interface are calculated and the building surfaces are calculated. The building surface locations are adjusted accordingly.

Mesh regeneration: The mesh of the fluid domain is regenerated to accommodate the displacement of the moving structure. The mesh geometry is updated and the mesh velocity fields is calculated for input to the CFD loop.

At each time step, a convergence tolerance of 0.0005 was used for the velocity and displacement norms. The convergence tolerance of 0.001 was used for the fluid-structure surface norm. All solutions were started with the entire domain at zero velocity and displacements. The iteration will finished when the steady state conditions were reached.

Results and Discussion

The test case presented in Part 3 is used to study the vortex shedding behind a building, the oscillation frequency of the structure and the fluid-structure interactions.

The frequency of vortex shedding of a bluff body is defined as follows in term of Strouhal number (Blevins 1990):
where $f_{vortex}$ is the frequency of vortex shedding, $S_t$ is the Strouhal number, $B$ is the characteristic length. The Strouhal number depends on the geometry and the Reynolds number of the flow. The Reynolds number of this test case is set to 4000 and the viscosity is adjusted based on inflow mean velocity and the square cylinder side length to match the Reynolds number. For uniform flow, the fluid simulation gives the value of 0.163 for Strouhal number which agree well with the result given by Blevins (1990).

Figures 4(a) and (b) represent the time history of the drag and lift coefficients acting on the square building, respectively. Figure 5 (a) and (b) shows the power spectral density of the alongwind (drag) forces and acrosswind (lift) forces, respectively. The mean value of the drag force coefficient is 1.40 which is 26% higher than the drag coefficient in Blevins (1990). The drag coefficient in the literature is for three-dimensional fix square cylinder. Because of the lack of experiments concerning flow induced vibrations of the flexible building, a comparison between numerical results and measurement data was unfortunately not possible. The mean lift coefficient is zero due to the building symmetry.

\[
f_{vortex} = S_t \frac{U}{B}
\]
The simulations start from stationary state and then start to move under the influence of vortex shedding. Figures 6 and 7 show the time history of the structural displacements. After several seconds a statistically steady flow is found. The periodic vortex shedding is established. The displacement of a tip point of the structure along the wind direction is presented in Figure 6. Firstly, the displacement amplitude increases quickly, then have a rapid decrease. Slowly, the steady flow state is approached. Figure 7 illustrates the time history of the structural displacements normal to the flow direction.

The time history of building displacement and the corresponding velocity for a tip point is shown in Figure 8. It can be seen that when the peak of displacement is reached the velocity of the building motion is close to zero. When the velocity of the structural motion is at its peak point, the value of the displacement is near zero.
Conclusion

In this paper, a coupled algorithm for the numerical simulation of fluid-structure inactions is developed based on the loosely coupled method. A partitioned coupling algorithm applies computational fluid dynamics method and structural analysis program for investigating fluid-structure interactions. Transient simulations were used to show the details of the interaction between the air flow and the tall building.

The segregate iterative procedure presented in this study provides an effective tool to couple fluid-structure interactions. The computing method can predict the response of the building with accuracy in the preliminary design stage.

The efficiency of different turbulence models, such as large eddy simulation models, will be investigated in the future work. For the structural simulation, 3-D finite element formulations should be supplemented. Another main aspect of our future research will also focus on improving the efficiency of the current algorithm.
Acknowledgements

This research is financially supported by the National Natural Science Foundation of China(50678122, 50621062, 90715040 and 50708014) and National Technology R&D Program of China (2006BAJ06B05). Our gratitude also is expressed to Shanghai Postdoctoral Foundation(06R214153), the National Postdoctoral Foundation of China(20060390164) and Shandong Jianzhu University doctoral Foundation.

References


