A STUDY ON NUMERICAL SIMULATION ON FLOW-FIELDS & WIND-INDUCED NOISE AROUND BUILDINGS

Young-duk Kim¹, Yohanes Joni Tri C² and Seung Jeong Baek³

¹ Professor, Environmental Engineering, Kwandong University, Korea, kimyd@kd.ac.kr
² Researcher, Environmental Engineering, Kwandong University, Korea, joni@kd.ac.kr
³ PhD candidate, Environmental Engineering, Kwandong University, Korea, baeksj04@naver.com

ABSTRACT

As residential buildings in urban areas are getting higher, there are many complaints about wind-induced noise generated by strong wind. People living in surrounding high-rise buildings have been expressed their fear of being disturbed by wind noise. This has become one of environmental problem to be solved. However, there are many difficulties in applying experiments on wind-induced noise problems. Those problems mainly caused by the following factors: (1) difficulties to distinguish between the natural wind and wind-induced noise, and (2) model scale’s problem for Strouhal number and Reynolds number. Nevertheless, the effort to study and reduce on the wind-induced noise around building is still rare in Korea or even in the world. Previous researches related with wind noises were mostly focused on aviation and machinery field. In this study the flow-fields are obtained from Large Eddy Simulation (LES) Smagorinsky turbulence model, and the wind induced noise predictions are computed using SYSNOISE Rev5.6Beta. The results were compared with the experiment. The comparison between experiment results and numerical simulation results showed the very similar trends for peak SPL were exhibited in various cases. In this study, Computational Aeroacoustic is known as a very useful tool mean on wind-induced noise for buildings. The researches to compare in full scales and other models are needed in the future.

KEYWORDS: WIND-INDUCED NOISE, CFD, COMPUTATIONAL AEROACOUSTICS, SOUND PRESSURE LEVEL, AND LES.

Introduction

Recently, high-rise buildings were constructed in advanced country like Korea, Japan, USA and European country. As residential buildings in urban areas are getting higher, people living in surrounding high-rise buildings have been expressed their fear of being disturbed by wind noise. This has become one of environmental problem to be solved. However, there are many difficulties in applying experiments on wind-induced noise problems. Those problems mainly caused by the following factors: (1) difficulties to distinguish between the natural wind and wind-induced noise, and (2) model scale’s problem for Strouhal number and Reynolds number. Nevertheless, the effort to study and reduce on the wind-induced noise around building is still rare in Korea or even in the world. Previous researches related with wind noises were mostly focused on aviation and machinery field. On the one hand, there is a social (and economic) requirement for building constructions so that building functions can be generated without harm to the environment and hence also to humans. On the other hand, there is an individual need for quiet and peace in the home environment. Both these demands have to be considered on designing of building in the future.
Methodologies

The approach taken here to calculate the noise radiation from the cylinder is based on a generalized form of the Lighthill acoustic analogy (Ffowcs Williams & Hawkings). In this analogy, the Navier-Stokes equations for the flow are re-written as:

\[
\frac{1}{c^2} \frac{\partial^2 p'}{\partial t^2} - \frac{\partial^2 p'}{\partial x_i \partial x_j} = \frac{\partial Q}{\partial t} - \frac{\partial F_i}{\partial x_i} + \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j}
\]

Where \((p')\) is the acoustic pressure fluctuation, \((Q)\) the mass flow rate, \((F_i)\) the external forces acting on the fluid and the Lighthill stress tensor containing momentum flux, thermal and viscous terms and \((C)\) the speed of sound. The source terms on the right hand side of the equation (1) (essentially the monopole, dipole and quadrupole fluctuations) are obtained from a transient simulation using CFX-11, with a LES turbulence model. The CFD provides time series of the dipole (pressure fluctuation on the models surface) and the quadrupole (essentially in the wake) sources. The acoustic analogy equation is solved in the frequency domain with SYSNOISE rev5.6 beta, using the Direct Boundary Element Method (DBEM) model. Figure 1 show the methodology of CFD simulation and process for wind-induced noise.

Outline of CFD analysis

Three types of columns (circular, rectangular and triangular), were placed in the turbulence boundary layer. Figure 2 shows the cross sections, dimension of columns and wind direction. The computational domain covers the following: 30d (X1 direction), 20d (X2 direction), and 10d (X3 direction).

![Figure 2 Cross sectional shapes, dimension of columns and wind direction.](image)

This domain was discretized into 50 (X1) x 40 (X2) x 20 (X3) meshes for the case of LES. The value of mesh interval adjacent to solid wall was set at 1/24d for all cases see figure 3.
This domain was discretized into $50 \times 40 \times 20$ meshes for the case of LES. The value of mesh interval adjacent to solid wall was set at $1/24d$ for all cases. A hexahedral mesh has been used with a total number of elements of 76000. To keep the LES tractable, the spanwise extent of the computational domain has been kept relatively small and periodic boundary conditions have been imposed in the spanwise direction.

In order to obtain better agreement between experimental and numerical results, boundary conditions adopted in the numerical simulations should be the same as those in the experiments, especially for inflow boundary conditions. The magnitude of the Inlet velocity is specified and the direction is taken to be normal to the boundary, because of the velocity profile in the wind tunnel test was described as uniform flow. In the simulation on inlet boundary conditions, subsonic flow was setup as the flow regimes, the total mass flow rate into the domain was setup as normal speed (20 m/s), In both of side walls was setup as free-slip conditions and the columns model where located in the bottom side was setup as no-slip condition. Boundary conditions are summarized in Table 1.

<table>
<thead>
<tr>
<th>Location</th>
<th>LES (settings)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inflow</td>
<td>Flow regime = Subsonic, Normal speed = 20 m/s, Turbulence Intensity 20% (4m/s)</td>
</tr>
<tr>
<td>Outflow</td>
<td>Flow regime = Subsonic, Mass and momentum = Average static pressure, Relative pressure = 0 Pa, Pressure averaging = Average over whole outlet</td>
</tr>
<tr>
<td>Wall</td>
<td>Wall influence on flow = Free slip</td>
</tr>
<tr>
<td>Model</td>
<td>Wall influence on flow = No slip, Wall roughness = Smooth wall</td>
</tr>
</tbody>
</table>
Result and Discussion

The velocity results of CFD calculations are shown in the horizontal symmetrical cross section with wind direction as shown on figures 4. It shows the distributions of time average velocity field (m/s) at horizontal sections. The three types of columns were placed in a cross section. As shown on the figures 3, the high velocity is represented in dark colors while than light colors represent lower velocity. It shows that, the maximum speed of circular shape is higher than rectangular and triangular.

![Velocity Distributions](image)

Figure 4 Velocity (m/s) distributions at horizontal section of three columns

Fig. 5 illustrates the distributions of time average pressure field (Pa) at horizontal symmetrical cross-section with wind direction shown below. Under this condition, it is shown that the pressure fields show on various styles by shapes. For the circular shape, it shows that lower pressure in the region beside the model as represented in the light colors, it observed that low pressure region that mean higher velocity region. For the rectangular the lower pressure can be seen in region near the corner, while for triangular can be seen in the region near the angle corner region. Thus pressure distributions by building shapes showed in order to circular > rectangular > triangular shapes building.

![Pressure Distributions](image)

Figure 5 Pressure distributions at horizontal section of three columns

**Computational aeroacoustics (CA)**

1. Measuring system.

As well as in the experiment, for the measuring system on simulation, the measuring point can be determine at 500 mm away from the shape of columns as shown in figure 6. So than the SPL radiated can be measured in that distance. In the computational aeroacoustics the point measurement can be calculated at process field point as the measuring place.
2. Transferring data input.

Figure 7 show the distribution of velocity versus time value from the calculation result of CFD. The calculation has done with 10,000 time steps, than the result has been chosen in every 3-time steps and the transfer data input of 128 res. files from the end of the time step results has been used to calculate the Sound Pressure Level. The reason of this is because the velocity variation is more stable in that range.

3. SPL fluctuation and peak SPL on CA results.

In this section, the fluctuating sound sources calculated by CFD have been transformed in the frequency domain to produce distributed sound sources on the columns body. The radiated field from the sound sources was calculated using the DBEM model in SYSNOISE. The result of understanding the ears sensitivity gives us a reverse curve, which is used for the dBA measurements. By using this curve as a correction value to a flat measurement, dBA measurements more closely approximate how humans will perceive the sound or noise them hear.

The calculated SPL at various frequencies at measuring points’ location is plotted in Figure 8. For the comparison of computation in each type, and measuring points has been chosen to calculate sound pressure level (dB A) at horizontal sections.
Figure 8 SPL by frequency of computation and experiment results at measuring point (500 mm away from the shape of columns).

The calculated SPL at various frequencies at measuring point (500mm away from shape of columns) is plotted in figure 11. In this comparison we consider to compare only in the peak of SPL between computation and experiment for three types of columns (circular, rectangular and triangular). It clearly shows from the computation and experiment results, the maximum SPL was 90dBA at the frequency 410Hz (computation) and 83dBA (experiment) in circular columns, 83dBA at frequency 400Hz (computation) and 77dBA (experiment) in rectangular, and 80.5 dBA at frequency 420Hz (computation) and 73dBA (experiment) in triangular. so the highest peak of SPL in order of circular column > rectangular column > triangular column.

4. Comparison of overall SPL.

The overall SPL from the CFX-11-SYSNOISE coupling has been compared in figure 9. In this comparison we consider to compare only in the peak of SPL from experiment for three types of columns (circular, rectangular and triangular). The prediction obtained from the CFX-11-SYSNOISE coupling overestimates the peak of SPL from the experiment results by about 7 dBA in circular, 6 dBA in rectangular and 8 dBA in triangular columns. The comparison was made from the experimental and computation results in the peak SPL of 3 building shapes. It shows that, the peak SPL (dB A) by frequency distributions of calculations
and experimental results have good agreement on the dimensional distributions, it shown the trend of SPL distribution are similar trend to the experimental results. And the peak SPL by building shapes showed in order to circular > rectangular > triangular shapes building. About some overestimation between computation and experiment it might be the difficulties in wind attack angle in the experiment because it might be different angle can also produce different Sound Pressure Level,

![Figure 9 Comparison of overall SPL from computation and experiment results](image)

**Conclusion**

In case of the building model, physical model experiments on wind-induced noise are very difficult to do full-scale because of problems in meeting both Re. and St. Numbers at the same time scale. Thus, the various models for building were conducted for wind-induced noise on three building shapes (circular, rectangular and triangular) in a cross flow. And the results were compared with the experiment. The result of LES reproduce well in the flow features (velocity and pressure distributions) has related to the SPL around buildings is shown different results by the shapes. It means that the higher velocity and pressure distribution around building shapes is also the higher SPL can be obtain. The peak SPL by building shapes showed in order to circular > rectangular > triangular shapes building.

In this study we know that Computational Aeroacoustic is very useful tools mean on wind-induced noise for buildings, and in the future, it is need to continue this research to compare in full scale's and other models.

**Acknowledgment**

The work described in this paper was partially supported by SOC Project (05-GIBANGUCHUK-D03-01) through the Design Criteria Research Center for Abnormal Weather-Disaster Prevention (DCRC-AWDP) in KICTTEP of MOCT, KOREA and RIC(N) of Kwandong University

**References**

Kim Young-duk, "Experimental and Numerical Study on Wind-Induced Noise of Thin Columns" The Proceedings of 4Th International Conference on Advances in Wind and Structures (AWAS'08), p195-200, 2008


Osamu Mochizuki, "Introduction to vortex noise", 2002


Montavon, C, 2001, Setting up aLES model in CFX-5.5, ALESSIA Deliverable D1.4.