Numerical Analysis of a Blast Wave Using CFD-CAA Hybrid Method

In Cheol Lee* and Duck-Joo Lee.†
Korea Advanced Institute of Science and Technology, Daejeon, 305-701, Republic of Korea

Sung Ho Ko‡ and Dong Soo Lee§
Chungnam National University, Daejeon, 305-764, Republic of Korea

and

Guk Jung Kang**
Agency for Defense Development, Daejeon, P.O. Box35, Republic of Korea

CFD (computational fluid dynamics)-CAA (computational aeroacoustics) coupled methods are developed for the analysis of blast noise. Using CFD methods and CAA methods, a distributed monopole problem is simulated and the results are compared to each other. These results show the difference between CFD methods and CAA methods, and give the need of CFD-CAA coupled methods. The developed CFD-CAA coupled methods are applied to analysis of supersonic shock noise for validation. From the results, the developed methods can be used for not only simple cases but also complicated practical cases. After validation, the developed methods are applied to the problem of real high pressure condition and compared with experimental data. For near field flow analysis, the realistic condition of high pressure is simulated without projectile effect. Using a silencer of simple type, we can reduce maximum pressure of blast noise up to 76%. And the effect of key parameters such as diameter, length of a silencer and number of baffle are investigated. For noise analysis, noise of near field and far field is calculated both with silencer and without silencer. From the noise directivity map, we can know the characteristics of a blast noise and its silencer.

Nomenclature

\[
\begin{align*}
D_N & = \text{diameter of barrel exit} \\
D & = \text{diameter of silencer} \\
L & = \text{length of silencer} \\
\rho & = \text{density} \\
x & = \text{position in the axial direction} \\
y & = \text{position in the radial direction} \\
u & = \text{velocity in the axial direction} \\
v & = \text{velocity in the radial direction} \\
p & = \text{pressure} \\
p_\infty & = \text{ambient pressure} \\
R & = \text{distance from barrel exit to data acquisition point} \\
\theta & = \text{angle between } R \text{ and } x\text{-axis (axial direction)} \\
f & = \text{frequency} \\
M_S & = \text{Mach strength}
\end{align*}
\]

* Ph.D. student, Aerospace Engineering, essence@acoustic.kaist.ac.kr, AIAA student member.
† Professor, Aerospace Engineering, djlee@kaist.ac.kr AIAA Member.
‡ Professor, Mechanical Design Engineering, sunghoko@cnu.ac.kr.
§ Master student, Mechanical Design Engineering, free-ds@hanmail.net
** Research fellow, Agency for Defense Development, kshsj85@hanafos.com
I. Introduction

When a gun fires, there exists excessive noise which propagates as a form of blast wave. It can be known that the noise from a gun gives serious damage to structures and it effects human bodies and environments. This effect causes both social and military problems. So, it is important to understand the characteristics of blast waves and derive some ideas for reduction of noise.

Recently, as computational performances increase, the methodology of computational fluid dynamics (CFD) has developed rapidly. In early nineties, CFD methodology can provide the analysis results of one-dimensional laminar flows, but nowadays it can analyze most of flow cases. And methodology of computational aeroacoustics (CAA), which can simulate both flow and noise simultaneously, also has been developed. Generally, amplitude of acoustic disturbance is much smaller than that of flow and has a characteristic of high wave number. So, CAA methodology, which can analyze acoustic noises precisely, has characteristics of high-order and high-resolution.

CFD method can solve complex geometries or conditions in a short time. But it does not have characteristics of high-order, and high-resolution relatively. For example, a noise source of 100 dB is actually the same as 0.01 percent of atmosphere pressure. For successful analysis of acoustic noise, the solver should have precision of millions. CAA method has precision of millions and can provide high-order, and high-resolution results, but it has some limitations for complex geometries and it needs too much time, relative to CFD method.

Up to the present, the analysis of aeroacoustic noise has been made by two ways. First, theoretical equation and empirical formula are used for the analysis of aeroacoustic noise. These methods are efficient since they can be easily applied, but it is hard to understand the detail characteristic of noise field, such as directivity by distance. Second, flow analysis and acoustic analogy can be used. Using these methods, we do not need to have a new experiment when there are some changes of conditions and geometries. Through flow analysis, we can expect the influence of changes without additional experiments. But if there is a moving body in domain, it can be difficult to apply on. We have to use Ffowcs Williams and Hawkins equation for moving bodies, but in equation, there exists quadrupole term and it only can be calculated by real-time volumetric integral which is almost impossible. Additionally, acoustic analogy can not be applied on when there exist shocks or supersonic region. Therefore, CFD-CAA hybrid methodology, which has merits of both CFD and CAA, is suggested and applied on supersonic blast noise.

Before applying hybrid method, the difference between CFD results and CAA results are observed through analysis of monopole wave propagation problem. After that, the hybrid method is applied to monopole wave propagation to verify the developed method. Finally, the CFD-CAA hybrid method is applied to a realistic blast wave case and realistic silencer attached cases.

II. Methodology and numerical schemes

A. CFD-CAA Hybrid Method

The basic concept of CFD-CAA hybrid method is to take merits of both CFD method and CAA method. CFD method has a merit that it can deal with complicated geometries and conditions in a short computational time. CAA method has a characteristic of high-order, and high-resolution which yields precise results. In hybrid method, CFD method simulates complicated near flow field regions which contain complicated geometries of nozzle exit and silencers. CAA method simulates far acoustic field regions which have small amplitude of pressure perturbation.

Figure 1 shows a schematic draw of CFD-CAA hybrid method. First, whole computational domains are analyzed by CFD method. In CFD analysis, some data, velocity, density, and pressure, are acquired as time-dependent terms along a data transfer line (dashed line), which is the first line of CAA computational domain. After that, the outer region of data transfer line is analyzed by CAA method and the time-dependent terms are used as boundary conditions.
B. Governing equation and Numerical Scheme

The governing equation which is used for analysis of blast wave can be written as below.

\[
\frac{\partial \hat{Q}}{\partial t} + \frac{\partial \hat{E}}{\partial \xi} + \frac{\partial \hat{F}}{\partial \eta} + \frac{\partial \hat{G}}{\partial \zeta} = \hat{S},
\]

(1)

Navier-Stokes equation, which considers viscous term of right-hand side, is used for CFD analysis. Compressible Euler equation, which makes right-hand side of equation (1) goes to zero, is used for CAA analysis.

CFD method uses Roe’s upwind scheme as spatial derivative and 2nd order central scheme for viscous term. LUSGS implicit scheme is used for time integration and turbulent model is used. Additionally, chimera grid technique is used for easy analysis of complicated physics around nozzle exit. CAA method uses 4th order optimized high order compact (OHOC) scheme [1] as spatial derivative and low dissipation and dispersion Runge-Kutta (LDDRK) scheme [2] as time integration. These two schemes have high-order, and high-resolution characteristics, which minimize dissipation and dispersion errors. Also, adaptive nonlinear artificial dissipation (ANAD) model [3], and generalized characteristic boundary conditions [4] are used for physical results.

III. Numerical results

A. Distributed monopole problem

To verify the CFD-CAA hybrid method, distributed monopole problem is selected. Before applying hybrid method, the problem is solved by each CFD and CAA method to compare the results of both methods and to understand a difference between CFD method and CAA method. The computational domain is shown in Figure 2. The governing equation can be written as equation (2), and the right-hand side term represents a source term by monopole.

\[
\frac{\partial \hat{Q}}{\partial t} + \frac{\partial \hat{E}}{\partial \xi} + \frac{\partial \hat{F}}{\partial \eta} = S, \quad S = [\hat{m}, \hat{mu}, \hat{nv}, \hat{ne}]^T
\]

(2)

where \( \hat{m}(x, y) = 10^{-4} \exp \left\{ -0.05(\Delta x^2 + \Delta y^2) \right\} \sin(2\pi f t) \)

In equation (2), \( \Delta x \) and \( \Delta y \) mean a distance in x and y direction from point of \((x, y) = (0,0)\), which is the location of monopole source. The cases of \( f=0.05 \) and \( f=0.1 \) are investigated. The all grid spaces in monopole problem are equally 1, so \( f=0.05 \) means twenty grid points for a wave, and \( f=0.1 \) means ten grid points for a wave. In this part, 2nd order central scheme is selected for CFD scheme, and 4th order OHOC scheme is selected for CAA scheme.

For the case of \( f=0.1 \), we can observe more significant difference between CFD and CAA calculations. Figure 3 shows the comparison of pressure contours for the case of \( f=0.1 \). CAA scheme simulate monopole without deformation, but CFD scheme distorts the form of monopole and has some effect of dissipation. The comparison of pressure signal gives more clear understanding of difference between CFD result and CAA result. From figure 3, we can observe that CFD calculation yields amplitude error in the near region of source part. And as x becomes larger, CFD result shows not only amplitude error but also phase error, where as CAA result shows good agreement with exact solution. (figure 3-b).
We can know that if there are sufficient grid points, CFD calculation and CAA calculation have little difference in the region of near field, but show some difference in the region of far field. If the grid points are insufficient, CFD calculation shows less precise results than CAA calculation in the region of both near field and far field. So, near field region can be simulated by CFD method, which has little difference with CAA method in near field, and far field region should be simulated by CAA method, which yields better results.

B. Validation of hybrid method

At previous section, we applied CFD-CAA hybrid method to a simple monopole problem. Now to check the stability of developed method, the hybrid method is applied to shock tube problem done by Wang and Widhof [5]. The computational domain is shown in figure 13. Dimension of length is non-dimensionalized by diameter of nozzle. Total number of grid point is about 35,000. Initial conditions at pressure discontinuity are $p_2/p_1=3.47$ and Mach number of shock is set to 1.76, which are the same conditions in real experiment.

First, whole computational domain is simulated by CFD method. During CFD analysis, primitive variables with respect to time are obtained along line of $y=0.8$, a horizontal line in figure 4. After that, the domain above the line of $y=0.8$ is simulated by CAA method. The primitive variables are used as boundary conditions at line of $y=0.8$, which is bottom line of CAA’s computational domain. As mentioned in the section of numerical scheme, Roe’s upwind scheme, 2nd order central scheme and LU-SGS scheme are used for CFD scheme, and 4th order OHOC scheme and Runge-Kutta scheme are used for CAA scheme. Since CFD method and CAA method use different numerical schemes, the time step of each result is different. So, the time step and primitive variables with respect to time from CFD calculation can not be used directly to CAA analysis. Some treatments such as interpolation are needed to apply primitive variables as boundary conditions in CAA analysis.

Through calculation, the pressure variation with respect to time is obtained and compared with other results in figure 5. From figure 5, we can know that CFD-CAA hybrid method works well. CFD-CAA method and fully CFD method show similar result, but variation of CFD-CAA result
is more sensitive and clear with respect to time than fully CFD result, which shows some dissipation effect. Like the problem of distributed monopole problem, if the observation position goes further from nozzle exit, the CFD-CAA method may give more correct result than fully CFD method. From this simulation, we can conclude that the developed CFD-CAA hybrid method can be applied to real flow problems.

C. Analysis of blast wave

To simulate a blast wave from a tank gun, we assume the tank gun as a shock tube. And the inside tank gun nozzle is set to be filled with gases of high pressure and high temperature. Computational domain, initial conditions and boundary conditions for CFD analysis are shown in figure 6. The domain is scaled down by the diameter of nozzle exit. The thickness of nozzle wall is set to 0.25 times of nozzle diameter. Inviscid adiabatic wall condition is applied to all wall regions as boundary condition. At inflow and outflow boundaries, characteristic boundary condition is applied which is based on Riemann invariants. The pressure ratio at nozzle exit is about 852, and the Mach number of shock is about 27, which are acquired from experimental data. Grid
system consists of three blocks and total number of grid point is about 97,000, and grids are condensed around nozzle exit which has complicated flow situation and great pressure gradient by high pressure ratio. Information line for data acquisition of primitive variables with respect to time is located at y/D=0.8. (dashed line in figure 6)

A flow field of high pressure is divided into two parts. One is an early time shock wave generated by gas explosion of high pressure and high velocity. The other is a plume generated by expansion of gases inside nozzle. Figure 7 shows contours of Mach number and pressure at 0.1 ms and 0.3 ms. From Mach number contours, we can know that upper region of plume has larger Mach number than center line region. And a vortex ring which is induced at upper region of plume is stretched by plume as time increases. From pressure contours, we can see that plume has lower pressure, and early shock wave propagates with distorted circular form. At figure 7 (c) and (d), we can observe that there exists a protrusion around center line, which is called carbuncle problem. When a shock wave lies perpendicularly between two grid points, an instability error occurs and this error leads carbuncle problem. This error is a numerical error generated by multi-order Roe’s scheme, and it can be minimized with large artificial dissipation term.

CFD-CAA hybrid method is applied to real blast wave problem. In this problem, the pressure ratio is over 850, the condition is too severe to guarantee the stability of CAA scheme. To ensure stability of hybrid method, the coefficient of artificial dissipation in CAA analysis is changed. The result is shown in figure 8. In figure 8, σ represents the amount of artificial dissipation term, and the case of σ=0.25 is the original method. If σ becomes larger, the amount of artificial dissipations in CAA calculation becomes larger, and the CAA analysis has more numerical stability. When σ equals to 0.25, the original calculation, the CAA calculation diverges before non-dimensional time reaches 0.6. And some oscillations are observed. If σ increases to 1.00, the results show more stability, however there still exist some oscillations.

Figure 8. Density contours for the high pressure blast flow with respect to coefficient of artificial dissipation

The developed method was validated for the case of low pressure ratio problem. However the pressure ratio of real blast wave is more than eight hundred fifty times of ambient pressure. The pressure ratio is so enormous that the developed method may not assure the correct results. To confirm the validity of developed method for high pressure ratio, a time dependent pressure signal is acquired and compared with experimental data and ideal scaling model. The ideal scaling model is suggested by Kevin S. Fansler[6] and has been applied to several kinds of weapons and gives good results for most cases. Using ideal scaling model, we can get the highest peak value of pressure signal and the arrival time of blast wave at a certain point. The equations of highest peak value and arrival time for the case of 120mm canon can be expressed as below.

\[
\bar{P} = \frac{P}{P_\infty} = 0.11 \left( \frac{\lambda}{\gamma} \right) + 0.0061 \left( \frac{\lambda}{\gamma} \right)^2
\]

where \( \lambda = 26.514 \left( 0.78 \cos \phi + \sqrt{1 - (0.78 \sin \phi)^2} \right) \)

\[
\tau = 0.0176 \left( X - 0.064 \ln \left[ 2X + 2 \left( \frac{P}{P_\infty} \right) + 0.128 \right] - 0.1 \right)
\]

where \( X = \sqrt{\frac{\tau}{\tau}} + 0.128 \left( \frac{\tau}{\tau} \right) + 0.0074 \)

(3)
The results of numerical simulation and ideal scaling model are shown in figure 9. From the figure, we can know that the results of numerical simulation and ideal scaling model show good agreement but show some difference with experiment. Experimental result shows longer duration time than other results. But the highest peak pressure values of all results are almost same. From this result, we can see that the developed method can be applied not only problem of low pressure ratio, but also problem of high pressure ratio.

D. Analysis of expansive silencers

An expansive silencer is applied to real blast wave problem. The shape is simply cylinder and there are no baffles inside silencer. The schematic figure of computational domain, initial and boundary conditions are shown in figure 10. Everything, except an expansive silencer, is same as analysis of blast wave. The diameter of silencer is 4 times of nozzle diameter, and the length of silencer is 20 times of nozzle diameter. No baffles exist, and the diameter of exit region of silencer is 1.1 times of nozzle diameter. All wall thickness is set to 0.25 times of nozzle diameter. In calculation without silencer, the information line for CFD-CAA hybrid analysis is located at y/D=0.8, but in this calculation, with silencer, the information line is shifted to y/D=2.3 since there is an expansive silencer to y/D=2.25. The grid system consists of five blocks and total number of grid points is about 110,000.

Figure 9. Comparison of time dependent pressure signal at x=-5.78m, y=0.68m

Figure 10. Computational domain including initial and boundary conditions for analysis of expansive silencer

Figure 11. Solid rendering of pressure blast flow that is discharged in the expansion silencer at different times
Figure 11 shows pressure contours inside expansive silencer at different times. (a) and (b) show the situation that flow comes from nozzle and propagates downstream. (c) and (d) show the situation that flow reflects at front wall of silencer and propagates upstream.

Figure 13 shows the distribution of peak pressure by angle for various distances. All peak pressures are expressed in dB scale. From figures, we can observe that the silencer greatly reduces peak pressure especially when the angle is less than 30 degrees. If angle is bigger than 30 degrees, the reduction effect by silencer is remarkably reduced. In other words, the silencer reduces peak pressures greatly in direction of flow propagation. From figure 13-(b), we can also know that the silencer reduces peak pressure by more than 14 dB in the direction of flow propagation. Some data, such as data from 50, 60, and 90 degrees in figure 13-(b), are not acquired since the computational domain is not enough to cover those points.

![Figure 12. Schematic diagram of data acquisition point](image)

![Figure 13. Distributions of peak pressure by angle (θ)](image)

E. Simple parameter study of expansive silencer

<table>
<thead>
<tr>
<th>Diameter (Dₙ)</th>
<th>Length (L)</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
<th>Case 5</th>
<th>Case 6</th>
<th>Case 7</th>
<th>Case 8</th>
<th>Case 9</th>
</tr>
</thead>
<tbody>
<tr>
<td>4Dₙ</td>
<td>10Dₙ</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6Dₙ</td>
<td>20Dₙ</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8Dₙ</td>
<td>30Dₙ</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

American Institute of Aeronautics and Astronautics
Additionally to previous simulation, simple parameter studies of expansive silencer are investigated. Diameter and Length are set to key parameters of simple cylindrical silencers. Effect of baffles are not considered in this parameter study. All length scales are non-dimensionalized by the diameter of nozzle ($D_N$). The diameter is changed form $10D_N$ to $30D_N$, and the length is changed from $4D_N$ to $8D_N$. Totally, nine cases are simulated, and it is arranged at table 1. All simulation conditions are same as figure 10, except the dimensions of expansive silencer.

Peak pressures about volume of each cases are arranged in figure 14. The measure point is $x/D_N=2.121$ and $y/D_N=2.121$. Generally, the reduction rate of peak pressure is inversely proportional to the volume size of silencers. But for the case of $D=4$, reduction rate is proportional to the volume size. From the result of case 9, which is the largest volume size of all, shows the best performance for noise reduction. From figure 14, we can conclude that large size of silencer is effective to reduce peak pressure from blast wave. But large size of silencer means heavy silencer, which is ineffective for attachment. So it is desirable to improve silencer with other ideas, such as baffle structures.

**IV. Conclusions**

CFD (computational fluid dynamics)-CAA (computational aeroacoustics) coupled methods are developed for the analysis of blast noise. A distributed monopole problem is using hybrid method. The developed CFD-CAA coupled methods are applied to analysis of supersonic shock noise for validation. From the results, the developed methods can be used for not only simple cases but also complicated practical cases. After validation, the developed methods are applied to the problem of real high pressure condition. For near field flow analysis, the realistic condition of high pressure is simulated without projectile effect. Characteristics of near field flow are simulated by CFD method, and characteristics of acoustic field are simulated by CFD-CAA hybrid method. The effect of key parameters such as diameter, length of a silencer and number of baffle are investigated. For noise analysis, noise of near field and far field is calculated both with silencer and without silencer. From the noise directivity map, we can know the characteristics of a blast noise and effect of silencer.

**Acknowledgement**

This work is supported by Agency for Defense Development of Republic of Korea

**References**