

# COSMOS-V, an Aerodynamic Noise Simulator

Review

Nariaki Horinouchi

## Abstract

The present and future computational problems of the aerodynamic noise analysis using COSMOS-V, our in-house CFD software, are explained by focusing on the wind noise and the wind-throb phenomenon. In addition, the side-view mirror surface vibration is equivalently treated as an aerodynamic noise problem because of the similarity of their mechanisms in the sense that both phenomena are caused by flow fluctuations around an automobile body. In general, pressure fluctuations due to the aerodynamic noise are minimal compared to those of the flow field itself which generates the sound. To date, however, the present computational techniques cannot directly resolve the noise. Instead, in the present approach, the noise characteristics are often indirectly predicted by measuring the resolvable-scale fluctuation of the unsteady pressure field. Thus, the accurate

computation of the unsteady flow field is indispensable for a reliable aerodynamic noise analysis. In this regard, this paper presents three key computational techniques to attain accurate results using COSMOS-V. These include: 1. the overset grid method to generate the appropriate structured computational grid system in a complicated geometry; 2. the finite volume method (FVM) on the collocated grid system to conserve the mass and the momentum on the discretized fundamental equations; and 3. the weak compressible flow model derived through the assumption of a slight nominal density fluctuation to simulate the wind-throb phenomenon. Two computational results from COSMOS-V are shown for the side-view mirror surface vibration and the wind-throb phenomenon.

### Keywords

Aerodynamic noise, Wind noise, Wind-throb, COSMOS-V, Unsteady flow, Overset grid, Collocated grid, Weak compressible flow model

## 1. Introduction

With recent advances in computers, improvements in computational methods for calculating governing equations of flow field, and developments in automatic mesh generation methods, etc., numerical simulations of flow based on Computational Fluid Dynamics (CFD) for a number of automobile related problems have yielded computational results with practical and satisfactory accuracy within practical and satisfactory computing hours.

In some automobile related problems which include turbulence, turbulent heat transfer, and multi-phase flow, however, difficulties still remain in obtaining results with the requisite accuracy, even if much time is consumed for computation. One example of a difficult problem is fluid noise, which has drawn attention in recent years as automobile quietness has become more of an issue.

Fluid noise refers to noise induced from and occurring within the flow. Examples of fluid noise in automobiles are:

- (A) wind noise around the vehicle body, wind-throb,
- (B) noise created by engine's combustion, injection, and emissions,
- (C) noise created by the engines cooling fan, air conditioner fan, and the air conditioner vents, and
- (D) noise from cavitation in the oil pressure system.<sup>1)</sup>

Group (A) is often referred to by the general term "aerodynamic noise".

Present state and future problems of aerodynamic noise analysis using COSMOS-V, our in-house CFD software, will be discussed below.

---

## 2. Aerodynamic noise analysis

---

### 2.1 Aerodynamic noise

As mentioned above, aerodynamic noise, as defined in this paper, is the noise caused by temporal fluctuations of airflow around the body of a moving automobile.

Wind noise, one example of aerodynamic noise, is caused by fluctuations in vortices that occur around steps and protrusions. Wind noise can be further classified as noise created by a Karman vortex that occurs around long cylindrical objects such as antennae, and as noise created by three-dimensional separation vortices caused by steps such as the front pillar portion. The former is also referred to as "Aeolian noise", and because of the strength of the vortex's periodicity, the noise that occurs is a narrow band noise with a distinguished frequency. Meanwhile, the latter is referred to as a broad band noise, and does not exhibit a distinguished frequency. It exhibits the frequency characteristic of gradual attenuation with small occasional peaks over a broad range of 100 Hz to several kHz.

Wind-throb is the low frequency (approximately 10-50 Hz) noise that occurs within the vehicle compartment when the sunroof or side windows are open as the vehicle is in motion, and applies pressure on the ears of passengers. In regard to the sunroof a small device called a "wind deflector" prevents the wind-throb, so in actuality the phenomenon is seldom noticeable. Meanwhile, when driving with one side window open, the vehicle occupants will sense low frequency air vibrations that occur at certain speeds. The wind-throb is one type of the Helmholtz resonance, in which the vehicle

compartment acts as a resonance box.

Furthermore, when the vehicle travels at high speeds, fluctuations in the separation vortices around the side-view mirror cause the mirror surface to vibrate, and rear visibility is adversely affected. This vibration is called aerodynamic chattering vibration. In this paper, the vibrations are equivalently treated as aerodynamic noise because they both occur from fluctuations in air flow around the vehicle body, and the same analysis method applied as with wind noise.

### 2.2 Aerodynamic noise analysis

Fluid noise is recognized as a noise when the sound wave created by extremely small fluctuations in density of the flow passes through a uniform stationary medium and reaches the human ear. By solving the governing equation for compressible flow that expresses the behavior of flow density, in principle, the generation and propagation of the noise can be directly calculated. However, sound pressure fluctuations distinguishable by the human ear as noise have an intensity of only approximately  $10^{-5}$  that of flow pressure fluctuations. At present, because such tiny fluctuations are lost in calculation errors, it is not possible to conduct a direct simulation of fluid noise.<sup>2,3)</sup>

Accordingly, when conducting actual aerodynamic noise analysis, two methods can be applied after accurately calculating the fluctuations at the flow field that is the source of the noise: either the characteristics of the noise is indirectly predicted utilizing the flow pressure fluctuations, or the sound pressure at the point of observation is calculated applying the Lighthill-Curle theory, described later, to the computational results of the unsteady flow field.

For ordinary flow velocity of a vehicle in motion, the airflow around the vehicle body is treated as an incompressible flow, which allows changes in density to be ignored. However, in regards to wind-throb analysis, which is covered in detail later, the treatment of incompressible flow is inadequate.

The governing equations for dimensionless unsteady incompressible flow can be expressed as follows:

$$\frac{\partial u_j}{\partial x_j} = 0 \quad \dots\dots\dots (1)$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad \dots\dots\dots (2)$$

where

$$\tau_{ij} = \frac{1}{Re} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \dots\dots\dots(3)$$

Equation (1) is the equation of continuity, and equation (2) is the Navier-Stokes equation (momentum equation), where  $u$ ,  $p$  and  $Re$  represent speed, pressure and Reynolds number, respectively.

Also, the Lighthill-Curle theory determines the sound pressure  $P_a$  at any observation point using the following equation:

$$P_a = \frac{1}{4\pi c} \frac{x_i}{r^2} \frac{\partial}{\partial t} \int_S n_i P dS \dots\dots\dots(4)$$

where  $c$  is the sound speed,  $x_i$  is a component of the positional vector of the observation point,  $r$  is the distance to the observation point, and  $P$  is the flow pressure on an object surface. If the flow pressure on the object surface is determined at every time using equation (1) and (2), then it is possible to calculate the sound pressure at an observation point using equation (4). For example, it is reported that the wind noise induced from the flow field around a simple 2-dimensional cylinder can be accurately predicted using this method.<sup>4)</sup> As equation (4) can be derived from the governing equations for compressible flow by assuming the following ideal conditions and simplifying the equation, careful study is required to determine whether or not it can be applied as is to actual automotive problems.

- Unlimited space, in where the object is included completely.
- The distance to the observation point is sufficiently larger than the sound wavelength.
- The distance to the observation point is sufficiently larger than the size of the interior object.
- Flow velocity is significantly lower than sound speed.

---

### 3. Aerodynamic noise simulator COSMOS-V

---

In order to conduct a highly reliable aerodynamic noise analysis, it is essential to accurately calculate airflow fluctuations. In this section, a particularly characteristic computational technique used by COSMOS-V to achieve this objective will be explained.

#### 3.1 Overset grid method

Because of the accurate and efficient computations, a grid system called a "structured grid",

which is uniformly lined (orthogonal, equidistant) much like a chessboard at least in the computational space, is utilized in COSMOS-V. By changing coordinates, grid lines are not orthogonal or equidistant in the physical space. In each of grid cells, physical quantities (velocity, pressure, etc.) are calculated by the discretization of the basic flow equations based on a high-accuracy scheme described later. Generally, a grid called a "body fitted grid" is used, which fits the grid lines on the boundary surface of the target object and concentrates the grid points near the surface.

However, when geometries with complicated areas are computed, it is difficult to cover them with a single structured grid block of sufficient quality. Also, too much time and effort is needed to generate the grid, and the number of grid points increases.

To deal with such problems, a method called the "overset grid method"<sup>5)</sup> is introduced to COSMOS-V. This method focuses in local shapes on the object and boundary. After generation of a partial grid appropriate for each boundary shape or the characteristics of flow field, multiple grid blocks are layered over each other (so that data can be mutually transferred between the grids in the overlapped region) and the entire area to be computed is covered. With this method

- it is easy to handle complicated geometries, and
- grid changes can be reduced by using case studies, etc.

Not only this method is extremely effective for reducing man-hours and improving usability, but computational accuracy is improved as a result of the ability to generate a better grid.

An example of a 2-dimensional section of an overset grid used for computing the flow field around the body of a sedan is shown in **Fig. 1**. Various color-coded grids are used to cover the center, front end, and rear end of the body.

In some grids, physical quantities for the grid points on overlapping regions are given by the interpolation from other grids.

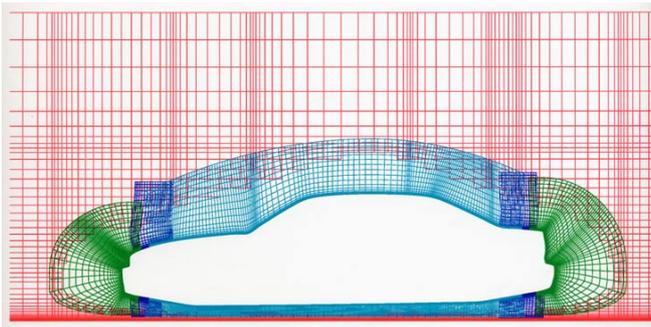
#### 3.2 High accurate discretization scheme

COSMOS-V uses a finite volume method based on a "collocated grid"<sup>6)</sup> to discretize the basic flow equation on the structured grid.

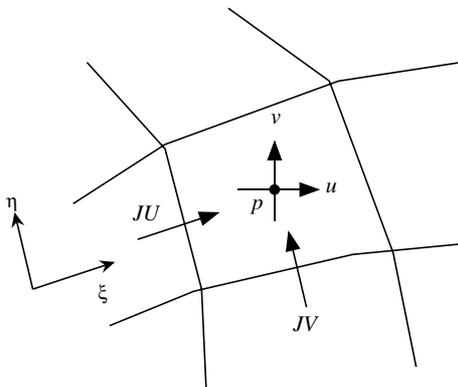
The finite difference method, which have been used in the past, adopts a "regular grid" that defines velocity component  $u_i$  and pressure  $p$  on the grid

points where the grid lines intersect. The previous version of COSMOS-V also used the regular grid. However, there are many problems with computational accuracy with this method. For example, the laws of conservation for physical quantities (law of conservation of mass, law of conservation of momentum) are not satisfied or the pressure fields oscillate.

Meanwhile, when the computational accuracy is emphasized, a "staggered grid" is generally used. A staggered grid is a discretization method that defines pressure  $p$  at the center of the grid cells and each velocity component on the cell interface in an orthogonal grid. The advantage of this method is that it accurately satisfies the laws of conservation. However, this method generates problems with its application to the actual computation. For example, it is difficult to expand into a generalized curvilinear coordinate system which is needed for a boundary fitted grid.



**Fig. 1** Example of overset grid system. (For simulation of flow around sedan type automobile)



**Fig. 2** Collocated grid on two dimensional plane.

In order to overcome above problem; COSMOS-V introduces a collocated grid that is capable of expanding into the generalized coordinate system, while also satisfying the laws of conservation as with the staggered grid. The collocated grid, while defining  $u_i$  and  $p$  at the center of the grid cells as shown in **Fig. 2**, also provides an auxiliary definition on the cell interface for mass flux  $JU_i$  which is interpolated from  $u_i$  using an interpolation method that is unique to the collocated grid.

The aforementioned governing equations (1) and (2) are converted by applying the coordinate conversion

$$d\xi_j = \alpha^{ji} dx_i, \quad \alpha^{ji} = \frac{\partial \xi_j}{\partial x_i} = \begin{bmatrix} \xi_x & \xi_y & \xi_z \\ \eta_x & \eta_y & \eta_z \\ \zeta_x & \zeta_y & \zeta_z \end{bmatrix} \dots\dots\dots (5)$$

to the following equations on the generalized coordinate system.

$$\frac{1}{J} \frac{\partial}{\partial \xi_j} (JU_j) = 0 \dots\dots\dots (6)$$

$$\begin{aligned} \frac{\partial u_i}{\partial t} + \frac{1}{J} \frac{\partial}{\partial \xi_j} (JU_j u_i) \\ = -\alpha^{ik} \frac{\partial p}{\partial \xi_i} + \frac{1}{Re} \frac{\partial}{\partial \xi_j} \left\{ (J\alpha^{mj}) \alpha^{mk} \frac{\partial u_i}{\partial \psi_k} \right\} \end{aligned} \dots\dots\dots (7)$$

where

$$J = \frac{1}{|\alpha^{ji}|}, \quad JU_i = (J\alpha^{ki})u_k \dots\dots\dots (8)$$

The Navier-Stokes equation (7) is solved for  $u_i$ , but the divergent term for velocity on the left side of the continuity equation (6) is evaluated using  $JU_i$ . By these careful discretization, it becomes possible to satisfy the continuity equation with high accuracy, while suppressing the pressure oscillations that arise in the calculations. Also, the convection term (the second term on left) of equation (7) is evaluated using  $JU_i$ .

For these governing equations, QUICK scheme (the third order upwind difference scheme) is applied to the convection term of equation (7), the second order accurate central difference scheme is applied to other space differential terms, and the Crank-Nicolson method is applied to the time

integration.

### 3.3 Weak compressible flow model

As noted before, the wind-throb is the phenomenon that occurs when Helmholtz resonance is induced within the vehicle compartment by the periodic vortex shedding at the opening of the sunroof or side windows. Because Helmholtz resonance is caused by slight density fluctuations, it is impossible to predict the wind-throb using a computation that assumes an incompressible flow.

Accordingly, the following governing equations, which models the weak compressibility on the flow field of low Mach numbers, are solved numerically in COSMOS-V.<sup>8)</sup>

$$M^2 \left\{ \frac{\partial p}{\partial t} + u_j \frac{\partial p}{\partial x_j} \right\} + \frac{\partial u_i}{\partial x_j} = 0 \quad \dots\dots\dots (9)$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} - u_i \frac{\partial u_j}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad \dots\dots\dots (10)$$

Here,  $M$  is the Mach number and has a value of approximately 0.1 at the flow around the vehicle body. The equations of the weak compressible flow is considered as the incompressible flow equation (1) and (2) with additional terms. In particular, the left side of the equation of continuity (9) expresses the effect of weak compressibility. Because the values are small, and also in order to accurately estimate the effects numerically, a method for accurately solving the original equation of continuity (1) is necessary. This condition is satisfied by utilizing the high accurate discretization scheme described in the previous section.

## 4. Computational examples

Below, two representative examples computed using COSMOS-V will be introduced. Please consult the references<sup>7, 8)</sup> for details.

### 4.1 Side-view mirror aerodynamic chattering vibration analysis

A case study using COSMOS-V was conducted, comparing an original visor geometry and an experimentally improved visor geometry for the side-view mirrors of a one-box shaped vehicle, testing whether or not differences in unsteady flow fields could be obtained.

The time averaged velocity vectors and pressure distribution ( $C_p$ : pressure coefficient) on the mirror surface yielded from the computational results are shown in **Fig. 3**. The original geometry shows large separation at the outside edge, as well as large

vortexes at the body side. It is also apparent that the area of low pressure on the mirror surface is more widely distributed with the original geometry.

Also, when fluctuations in pressure distribution on the mirror surface are animated, it becomes clear that the fluctuations are more severe with the original geometry.

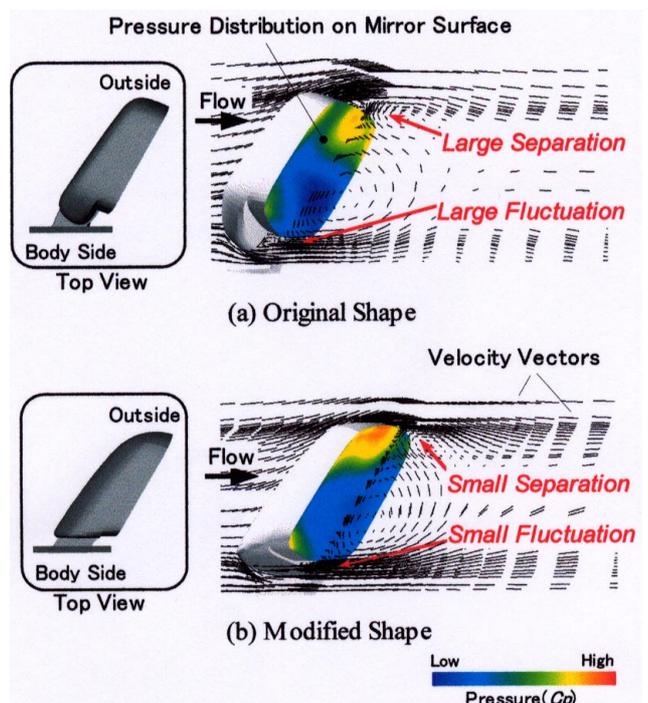
Taking the above mentioned information into account, it becomes clear that the pressure fluctuations on the mirror surface which cause the chattering vibrations become smaller due to the modified geometry's unstable flow field characteristics, such as smaller separation at the outside edge of the mirror and smaller vortexes at the body side.

### 4.2 Wind-throb analysis

For an example of wind-throb analysis, the computational results applied to a basic experimental model that was implemented to test the weak compressible flow model described in the **section 3.3** will be shown.

A rectangular box with an opening at its top is shown in **Fig. 4**. This is a basic experimental model called a three-dimensional open cavity, which simulates a vehicle compartment with an open sunroof.

The experimental results and the computational



**Fig. 3** Time averaged velocity vectors and pressure maps.

ones of the change in the wind-throb phenomenon when the flow velocity  $U$  above the cavity changes are compared. **Figures 5** and **6** show the changes in sound pressure level (SPL) and resonance frequency ( $f$ ) as related to flow velocity ( $U$ ).

In Fig. 5, both the calculation and the experiment equally presents the characteristic phenomena of such as wind-throb extremely high sound pressure levels at specific flow velocities. The dot-dash-line in the figure shows the results yielded by the computational method for the incompressible flow. In this case, the phenomena can not be simulated. The flow velocity at which the sound pressure level becomes the highest is also in agreement for the computation with the weak compressibility and experiment.

In Fig. 6, the frequency begins fluctuating up and down in relation to the flow velocity. It is thought that this kind of non-continuous change occurs when the mode (dot-dash-lines  $n=1, 2, 3$ ) for the frequency of vortex shedding changes as a result of Helmholtz resonance. The results of the calculations and the experiment are in agreement for this non-continuous change of frequency.

The thin horizontal line in Fig. 6 is the estimated value for the Helmholtz resonance frequency that is inferred from the area of the opening and the cavity volume for the basic experimental model. At a flow velocity of approximately 30 m/s, the frequency of the wind-throb seems to be pulled horizontally by the estimated value in a "Lock-in phenomena".

## 5. Conclusion

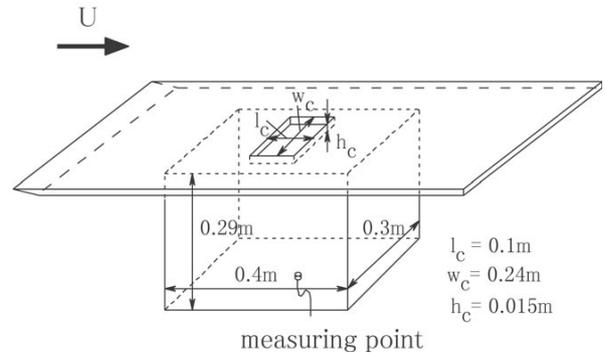
### 5.1 Present state

The current state of aerodynamic noise analysis has been discussed. With current computational techniques, it is impossible to directly simulate fluid noise. A realistic method is to (1) accurately calculate the unsteady flow field that involve the source of the noise, and then (2) estimate the sound pressure fluctuations from the pressure fluctuations of the flow field.

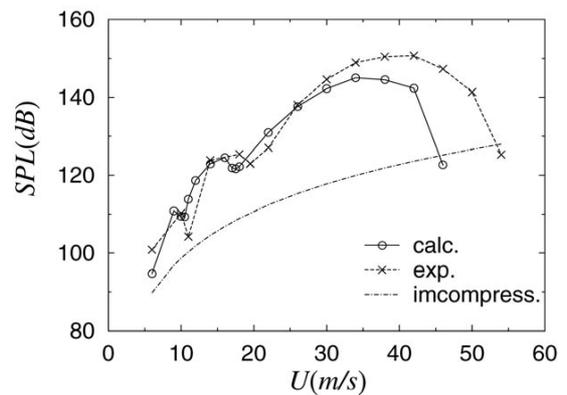
COSMOS-V is thought to be the most accurate and fastest in the world in regards to the aforementioned (1) calculation of the unsteady flow field. Also, COSMOS-V is the unique software which can predict the fluid-resonant noise such as the wind-throb phenomena.

Concerning the aforementioned (2) estimation of

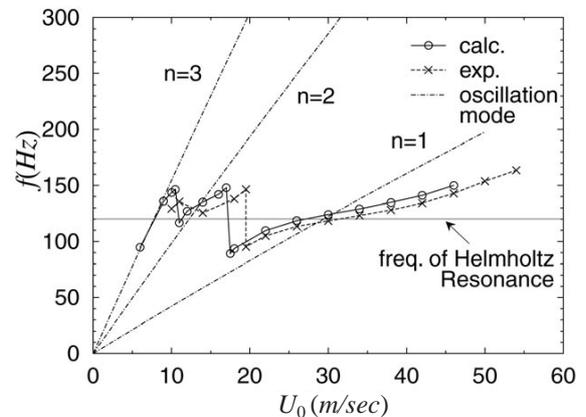
sound pressure fluctuations, Lighthill-Curle theory is thought to be the most reliable, but its field of application is limited. In such cases, it is necessary to use the pressure fluctuations of the flow field as a substitute.



**Fig. 4** Three-dimensional open cavity.



**Fig. 5** Sound pressure levels.



**Fig. 6** Frequencies of pressure fluctuations.

## 5.2 Future problems

There are still a number of technical problems remaining that must be resolved in order to move forward with the practical implementation of fluid noise analysis in the future.

Even with COSMOS-V, when predicting pressure fluctuations for high frequencies, accuracy drops with methods such as QUICK scheme which use a numerical viscosity. In the estimation of high-frequency pressure fluctuations, more accurate analysis techniques for unsteady turbulence, such as LES (Large Eddy Simulation), are necessary. COSMOS-V is developed with the expansion toward this type of method in mind, and more accurate predictions of unsteady flow fields are expected to be possible in the near future.

Also, concerning the aforementioned (2), a model with fewer limitations than the Lighthill-Curle theory is expected to be developed. Recently, our research has been progressing in that direction,<sup>9)</sup> but more time is needed for actual development.

One problem with computational techniques is how to increase the speed of unsteady flow calculations. The analysis of fluid noise requires numerous steps and the continuous calculation of the flow field, so a tremendous amount of calculation time is required. For example, when estimating the aforementioned wind-throb noise for a three-dimensional vehicle geometry, even with the super computer, which has a peak performance of 2GFLOPS, over 100 computing hours is needed for making calculations at each vehicle speed. Because numerous case studies are needed for the actual development of a product, increased calculation speed is required. In near future, it is thought that more emphasis will be placed on the use of parallel computers for faster calculations.

## Reference

- 1) Mochizuki, O. and Maruta, Y. : *Ryutai on kougaku nyumon* (in Japanese), (1996), Asakura Shoten
- 2) Kato, C. : *Turbomachinery*, **26-1**(1998), 17
- 3) Tani, I. : *Ranryu* (in Japanese), (1980), Maruzen
- 4) Ikegawa, M., et al. : *Bull. Jpn. Soc. Ind. Appl. Math.*, **6-1**(1996), 2
- 5) Kato, Y., et al. : *R&D Rev. Toyota CRDL*, **32-2** (1997), 23
- 6) Inagaki, M., et al. : *Proc. 10th Numer. Fluid. Symp.*, (1996), 410-411
- 7) Yamada, T., et al. : *2001 Jidousha Gijutsukai Chubu shibu Sokai Kenkyu Happyo kai Maezuri shu* (in Japanese), (2001), 79-82
- 8) Kato, Y., et al. : *R&D Rev. Toyota CRDL*, **36-2** (2001), 31
- 9) Kato, Y., et al. : *Proc. of Mech. Eng. Congr. 2000*, Vol.1, (2000), 951-952

(Report received on Oct. 4, 2001)




---

### Nariaki Horinouchi

Year of birth : 1961

Division : Applied Mathematics & Physics Lab.

Research fields : Numerical computational methods on aerodynamic noise analysis and the whole fields of CFD

Academic society : Soc. Ind. Appl. Mach., Jpn. Soc. Appl. Math. Inf. Process Soc. Jpn., Soc. Autom. Eng. Jpn.