Combined Experimental and Numerical Diagnostics for Near-field Flow around a Supersonic Flight Model

By Atsushi MATSUDA,1) Katsuya SHIMIZU,2)* Kakuei SUZUKI,2) Akihiro SASOH,2) Keiichi MURAKAMI3) and Takashi AOYAMA3)

1) Department of Mechanical Engineering, Meijo University, Nagoya, Japan
2) Department of Aerospace Engineering, Nagoya University, Nagoya, Japan
3) Japan Aerospace Exploration Agency, Chofu, Japan

(Received February 9th, 2011)

A system for evaluating the near-field pressure distribution around a supersonic flight model by combining experimental and numerical diagnostics has been developed. Experimental measurement is conducted using a ballistic range with four kinds of axi-symmetric flight models. Schlieren flow visualization is recorded using a high-speed framing camera and near-field pressure histories are measured using piezoelectric pressure transducers flush-mounted on the surface of flat plates in the test section. The numerical diagnostics is done using FaSTAR, a numerical simulation tool developed by the Japan Aerospace Exploration Agency (JAXA). The experimental and numerical data are compared to each other, and the numerical results well validated. Based on the numerical results, it becomes possible to estimate the accuracy of experimental conditions including the flight path and angle of attack, which cannot readily be determined only from experimental data, and to discuss the relationship between peak overpressure and aerodynamic performance. Satisfactory agreement between the experimental and numerical results at a flight Mach number of 1.66 ± 0.02 and important insights related to rear boom strength are obtained.

Key Words: Sonic Boom, Supersonic Flight, Ballistic Range, Numerical Simulation

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$r$</td>
<td>radial coordinate</td>
</tr>
<tr>
<td>$x$</td>
<td>axial coordinate originating in the head of the flight model</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
</tbody>
</table>

1. Introduction

In order to realize civil supersonic transportation (SST) for the next generation,1-3) serious sonic booms as well as poor fuel consumption efficiency remain to be crucial technical barrier.4) A high-accuracy boom evaluation technique is necessary for the development of advanced low-boom vehicles. In particular, an effective combination of experimental and numerical tools is a key for advancement. Recently, sonic boom evaluation methods are mostly based on numerical simulation. Therefore, an experimental study is expected to function as the validation process for the numerical simulation. First of all, the flow field near a test model (near-field) is predicted numerically. Then, based on the near-field pressure distribution, the wave propagation process is analyzed. Thus, it is a crucial issue to predict the near-field flow around test models accurately using numerical tools. Validation of the numerically solved near-field flow is necessary, which is done by a comparison with experimental results.

As experimental methods, actual flight and ground experiments are mainly conducted. However, an actual flight test is expensive, and the experimental opportunity is considerably limited. To the contrary, an experiment using ground test facilities can be conducted at low cost and with more frequent opportunities. For ground experiments, two types of facilities, wind tunnel and ballistic range, are used. The former is used more frequently than the latter because of its better availability.5-7) However, the effect of the sting, which supports the test model in a wind tunnel, needs to be carefully eliminated;8) results of even recent studies may have this problem.9) In a ballistic range, a sting is not necessary; a free-flight experiment is possible even in a modest-sized laboratory on the ground. It has a better function for measuring the pressure distribution around the whole test model, including the rear region. Although addressed much less so far,9-12) design guidelines for the rear configuration of free-flight vehicles warrants further investigation because the rear shock has an impact that is comparable to frontal shock.

© 2011 The Japan Society for Aeronautical and Space Sciences
* Currently, Kawasaki Heavy Industries, Ltd., Kagamigahara, Japan
Based on the above background, the combination of numerical simulation and ground test facility, especially the ballistic range, is expected to be a useful tool for sonic boom prediction.

The purpose of this study is to develop a useful system to accurately obtain the near-field pressure distribution around the whole free-flight body by combining a ballistic range experiment and numerical tools.

2. Method

2.1. Basic scheme

The scheme of the combination of experimental and numerical methods in this study is as follows.

In the experiment using a ballistic range, time variation of overpressures induced by the supersonic flight of a model is measured with piezoelectric pressure transducers flush-mounted on flat plates, and the density field around the model is visualized through the Schlieren method using a high-speed camera.

Numerical simulation for steady-state flight condition is conducted under the same conditions as those for the experiment, and overpressure variation is spatially traced along a line equivalent to a pressure transducer location. The numerical simulation is validated by comparing this numerically obtained overpressure variation with the experimental results. After this validation, parameters which cannot be obtained only in the experiment, for example, aerodynamic performance such as a drag, are output from the numerical simulation.

2.2. Experiment

In the experimental study, the square-bore ballistic range at Nagoya University is used. The launch tube has a 25-mm square bore. Combing the in-tube, aerodynamic sabot-separation technique, even a three-dimensional projectile can be launched without rolling.

Figure 1(a) shows the top view of the test section, and Fig. 1(b) shows the side view of the test section of the ballistic range. The pressure distributions around the test models are measured by piezoelectric pressure transducers (112A21, sensitivity: 7.8 mV/kPa, resonance frequency: 250 kHz, PCB Inc.). They are flush-mounted on flat plates installed in the test chamber. The sensors are set at 1500 mm from the muzzle exit. The distance from the designed flight path to a sensor is 150 mm. Even though a total of six pressure transducers (three transducers (left, center, right) in the lower plate and the rest in the upper plate) are used, one representative result from the sensor located at the lower center will be the focus of this study.

A baffle is used in order to mitigate muzzle blast, which is emanated from the muzzle exit prior to the launched projectile.

Schlieren visualization is conducted to capture the flow field around the projectile and the projectile attitude during free flight. The visualization system is also depicted in Fig. 1(a).

The flight speed of the projectile is measured by the “time of flight” method using two combinations of a diode laser and photodiode set with a 100-mm separation distance upstream of the pressure transducers.

2.3. Numerical diagnostics

Numerical simulation is conducted using FAST Aerodynamic Routines (FaSTAR) developed at JAXA. The calculation grid is automatically constructed using HexaGrid. Based on these programs, the three-dimensional Euler equation is numerically solved. The detailed description of the numerical procedures such as the flux discretization method, time integral method and so on are presented in Hashimoto et al.

3. Scheme Validation

3.1. Test models

In this experiment, two types of frontal surface configurations and two types of rear configurations were investigated. Hence, in total, four types of test models were launched. Those were symmetric models (X^0.75S and X^1.0S) and bluff models (X^0.75B and X^1.0B), respectively.

The X^0.75S and X^0.75B models are models proposed by McLean as low-boom generators that generate plateau-type pressure signatures. The radius of the cross-section is proportional to 3/4 the power of x, that is the cross-section area of this model is proportional to 3/2.
The surface configurations of $X^{1.0S}$ and $X^{1.0B}$ models are cones. Therefore, the radius of cross-section is proportional to 1.0 times the power of $x$.

Figures 2(a) and (b) show the symmetric models. Figure 2(a) is $X^{1.0S}$, and Fig. 2(b) is $X^{0.75S}$. The total length of the model is 125 mm (i.e., 50mm-long cone or $x^{3/4}$ part, 25mm-long cylinder part, and also 50mm-long symmetrical part) and the maximum diameter is 25 mm.

Figures 3(a) and (b) show the bluff models. Figure 3(a) is $X^{1.0B}$ and Fig. 3(b) is $X^{0.75B}$. The total length is 75 mm (i.e., 50mm-long cone or $x^{3/4}$ part and 25mm-long cylinder part). The frontal surface configuration is the same as the symmetric model (either $x^{3/4}$ or cone configuration.)

3.2. Comparison between CFD and experiment

Figures 4 (a) and (b) show the comparison of the flow field between the experiment and computational fluid dynamics (CFD) for the $X^{0.75S}$ model. The experimental result shown in Fig. 4(a) is based on the Schlieren method, and the CFD result shown in Fig. 4(b) is the pressure coefficient contour plot. In Fig. 4, the similarity of the flow field patterns between the experiment and CFD can be observed from the viewpoint of frontal shock waves, rear shock waves and expansion waves.

Figure 5 shows a comparison of the pressure signature between CFD and the experiment. For the plot of CFD results in Fig. 5, spatial pressure distribution along the line parallel to the body axis with the equivalent distance to the pressure transducer location is obtained from the steady-state solution. Then, the spatial distribution is converted to temporal distribution using the upstream flow velocity corresponding to the experimental flight velocity ($M=1.64$). It should be noted that the overpressure obtained from CFD should be corrected for comparison with the experimental overpressure due to the following reason. On one hand, the experimentally measured pressure is the value in the region behind the reflected shock wave at the sensor position. On the other hand, the CFD result is based on the static pressure distribution and the reflected shock wave at the sensor position is not taken into consideration. In this study, the overpressure due to the reflected shock wave is well approximated to be twice as high as that due to the incident shock wave. Therefore, the overpressure obtained from CFD is corrected to double the value for comparison with the experiment. From this figure, the frontal and rear peak overpressures agree between the experiment and CFD. The difference of duration time between the experiment and CFD is less than 10 µsec. Even though the pressure signature in the experiment in the region after rear shock wave is suffered from oscillation caused by the compression waves from the wake flow, the CFD result reproduces a pressure relaxation tendency. The correlation coefficient between the experiment and CFD in Fig. 5 is 0.953. Based on this CFD analysis, the experimental situation can be well reproduced quantitatively. Therefore, this CFD is used to diagnose the experimental flow field.
3.3. Evaluation of experimental flight conditions

In order to investigate the effects of flight conditions such as angle of attack and the distance between the flight path and the overpressure measurement point (flight path height) on the peak overpressure, parametric numerical simulations are conducted. Especially, a combination analysis between experiment and numerical simulation makes it possible to estimate the uncertainty of the angle of attack. This combination plays an important role since it is difficult to estimate the uncertainty from the visualization result itself due to the relatively coarse resolution.

The frontal peak overpressures versus Mach number with constant angles of attack (from –2 deg. to 2 deg.) are plotted in Fig. 6. This figure is the result for the X^0.75S model. The lines are from CFD analyses, and the symbols are from experimental results. Experimental conditions (Mach number and angle of attack) are also shown in Fig. 6. These conditions are estimated from the experimental results. Mach number is derived from the time of the flight method, and angle of attack is estimated from the visualization result. In the present measurement method, Mach number error originates from the measurement time error for the time of flight (±1 µsec). This time measurement error corresponds to a Mach number error of ±0.01. The error of the peak overpressure is estimated from the root-mean-square-error at the plateau region of the overpressure signature in Fig. 5. This root-mean-square-error is at most ±0.5 kPa. Even with this Mach number and overpressure fluctuation, the uncertainty of angle of attack is estimated within ±0.75 deg. from Fig. 6, in this experiment. From Fig. 6, the sensitivity of the angle of attack on the peak overpressure is 0.63 [kPa/deg.].

Fig. 6. Example of accuracy estimation for the experimental angle of attack; X^0.75S.

In experiment, an angle of attack of 0.75 deg. corresponds to only ±0.5mm deviation in the flight path height. Another possible reason for the deviation in flight path height is caused by the fluctuation of the launched model orbit (orbital fluctuation) from the designed flight path. The uncertainty in the flight path height is estimated at most ±2.0 mm from the flow field visualization in this experiment.

Figure 7 shows the typical relationship between the peak overpressure and the flight path height with the X^0.75S model calculated using CFD. As shown, the corresponding sensitivity is 0.03 [kPa/mm] at most. From Figs. 6 and 7, the effect of the uncertainty in the flight path height on the frontal peak overpressure is much smaller compared to the effect of the angle of attack.

Fig. 7. CFD results for the relationship between the frontal peak overpressure and flight path height; X^0.75S (M=1.66, α=0 deg.). Arrows correspond to the measurement uncertainty of flight path height in the experiment.

4. Results and Discussions

4.1. Frontal boom characteristics

Figure 8 shows the experimentally measured overpressure signatures for the X^0.75S and X^1.0S models. The difference of the frontal overpressure signature due to the nose shape is obvious. On one hand, the overpressure signature of X^1.0S has a spiky signature. On the other hand, that of X^0.75S has plateau one, being consistent to MacLean’s prediction.17) The frontal peak overpressure with X^0.75S is lower than that for X^1.0S by 40%. However, as for the rear shock wave, the X^1.0S and X^0.75S models yield similar overpressure variations. The duration time of the overpressure signature is of the same order as of τ, which is defined as the quotient of a model length to a flight speed; τ=221 µsec for X^1.0S and τ=224 µsec for X^0.75S. The characteristics of rear peak overpressure will be discussed in the next subsection.

Figure 9 shows the flow fields around the X^1.0S and X^0.75S models. The differences in the flow fields due to the model surface configuration can be seen. From the Schlieren images, the distance between the frontal shock wave and the expansion wave becomes shorter for X^1.0S as the distance from the test model increases, as shown in
Fig. 9(a). On the other hand, for X^0.75S, the frontal shock wave and the expansion waves lie almost parallel to each other, even with increasing the distance from the test model, as shown in Fig. 9(b). This difference is clearly observed in the CFD results, as shown in Figs. 9(c) and (d). For X^1.0S (Fig. 9(c)), after the frontal shock wave, the peak density region can be observed. On the other hand, for X^0.75S (Fig. 9(d)), in the region between the frontal shock wave and expansion wave, the density is distributed almost uniformly along the line parallel to the body axis. This flow field difference reflects the difference of the pressure signature in the nose region, as shown in Fig. 8.

4.2. Rear boom characteristics

Figure 10 shows flow field images over bluff models. The same comparison applies to the flow field around the front nose (i.e., frontal shock wave and expansion region) between X^1.0B (Figs. 10(a) and (c)) and X^0.75B (Figs. 10(b) and (d)). Yet, around the rear region, the differences between the X^1.0B and X^0.75B models are not straightforward. With both models, from Figs. 10 (a) and (b), a slip region behind the aftbody has a length equivalent almost to the forebody length; then, the wake follows. From CFD shown in Figs. 10(c) and (d), wake flow can also be observed. From Figs. 10(a) and (b), in the wake region, compression waves are generated, eventually leading to rear shocks. The duration time of the pressure signature becomes much longer than τ. See Figs. 11 and 12. Therefore, the wake plays a key role in the overpressure characteristics.
Figure 13 is the relationship between $\Delta P_{\text{rear}}$ and the duration time for the four types of models measured in this experiment. The circular symbols are $X^{0.75}$ models and rectangular symbols are $X^{1.0}$ models. The open symbols (circle and rectangle) represent bluff models, and the filled symbols represent symmetric models. The reason that the duration time is the same order between the bluff and symmetric models is due to the formation of the slip region behind the aftbody for the bluff models, as shown in Fig. 10. The increment in overpressure ($\Delta P_{\text{rear}}$) is lower for the bluff type ($X^{0.75B}$ and $X^{1.0B}$) than that for the symmetric type ($X^{0.75S}$ and $X^{1.0S}$), as suggested from Figs. 11 and 12.

The possible explanation for this is as follows. From the visualization results (Figs. 9(a), 9(b), 10(a) and 10(b)), the distance for compression waves to integrate into shock waves is longer for the bluff models (Fig. 10) than that for the symmetric models (Fig. 9). Based on this fact, wave intensities for the bluff models are expected to be weaker than those for the symmetric models. From the CFD results (Figs. 9(c), 9(d), 10(c) and 10(d)), this tendency is also suggested. The intensities of the waves in the rear region of the bluff models are weaker than those for the symmetric models. Moreover, while the rear shocks for symmetric models are clearly formed even in the near region of the rear edges, as shown in Figs. 9(c) and (d), the rear shock is not so clear in the wake region for bluff models (Figs. 10(c) and (d)). As a result, the rear shock waves for bluff models are weaker than those for symmetric models. The aftbody shape and associated wake significantly influence overpressure variation over a rear shock wave.
However, for the evaluation of total aerodynamic performance, the effect of $\Delta P_{\text{rear}}$ should be taken into consideration. Figure 15 shows the plot of the drag coefficient versus $\Delta P_{\text{rear}}$ with the same flow conditions as Fig. 14. The $\Delta P_{\text{rear}}$ of the bluff models (open symbols) are 0.7 kPa smaller than those of the symmetric models (filled symbols). On the other hand, the drag coefficients of the bluff models are 0.08 larger than those of the symmetric models. Therefore, by considering total performance including $\Delta P_{\text{rear}}$, selecting the symmetric or bluff model becomes a trade-off between low drag and low peak overpressure.

5. Conclusion

A system for evaluating near-field pressure distribution over a supersonic flight body is developed by combining a free-flight experiment and CFD diagnostics. For the scheme demonstration, four types of models (X$^\circ 0.75$S, X$^\circ 1.0$S, X$^\circ 0.75$B and X$^\circ 1.0$B) are utilized. The effects of the experimental flight conditions which are the orbital uncertainty and angle of attack on the frontal peak overpressure are assessed using parametric CFD diagnostics; thereby the uncertainty of the flight conditions in the experiment is evaluated. The frontal overpressure signatures for the X$^\circ 0.75$S and X$^\circ 0.75$B models are the plateau type, and the signatures for X$^\circ 1.0$S and X$^\circ 1.0$B are the conventional N-shape type. Wake effect has an important impact on the rear shock; the bluff models yield a smaller increment in the overpressure behind the rear shock ($\Delta P_{\text{rear}}$). Additionally, the trade-off between $\Delta P_{\text{rear}}$ and the drag coefficient is presented. As demonstrated, by combining a ballistic range experiment and CFD analysis, once the CFD results have been validated by comparison with experimental results, the system can even estimate aerodynamic coefficients, which are difficult to be directly measured during an experiment. It should be noted here that, for the present analysis of the near-field flow, the experimental data is essentially important to validate the CFD results. This combined system is expected to be useful, in particular, for further advancing the design of rear-boom alleviation.

Acknowledgements

The authors would like to express their gratitude to A. Saito and K. Yamamoto, Technical Division, Nagoya University for their valuable technical assistance. This research was supported by Japan Society for Promotion of Science as Grant-in-Aid for Scientific Research (S), No. 22226014, and by JAXA Funded Researches 27-J-J6710 and 27-J-J6774.

References