DSMC Simulation of Non-uniform Flow Effects behind a Conical Nozzle

Nobuyuki Tsuboi* and Yoichiro Matsumoto†

*Institute of Space and Astronautical Science, Japan Aerospace Exploration Agency, 3-1-1, Yoshinodai, Sagamihara, Kanagawa 229-8510, Japan
†Department of Mechanical Engineering, The University of Tokyo, 7-3-1, Hongo, Bunkyo-ku, Tokyo 113-8656, Japan

Abstract. The two-dimensional cylindrical DSMC simulations around a object were conducted in order to estimate the effect of the upstream condition. Non-equilibrium between translational and rotational temperatures at the nozzle exit is approximately 10K. The shock wave generated by the object and the boundary layer developed along the nozzle wall interact around the object. The comparison between the results with the nozzle exit flow and those in the uniform flow reveals that the temperature profiles around the object are influenced by the upstream flow conditions, and that the difference is approximately 20K.

INTRODUCTION

Experiments for an object in supersonic and hypersonic rarefied gas flow have been carried out for many researchers, however, freestream flow has to be generated by a convergent-divergent nozzle on the ground environment. It is well known that the frozen effects on the rotational and vibrational temperatures exist in the nozzle flow. The frozen effects are influence on the measured data such as a rotational temperature around a object, but the degree of the effects is unclear. Furthermore, the boundary layer in the nozzle becomes significantly thick because of low Re number in the rarefied flow. The order of Re is approximately $100 \sim 1000$, so that the boundary layer thickness is about 50% or more of the nozzle exit diameter. This thick boundary layer also affect on the non-uniform flow generated by the conical nozzle. However, the effects also have not been estimated in the past experiments[1].

Recently, numerical simulation is a predicted tool to calculate the flow for not only a continuum flow but also a rarefied flow. There exist some problems to evaluate non-equilibrium flow because of the lack of information for microscopic states. In the rarefied flow simulations, direct simulation Monte Carlo (DSMC) method is a useful method to evaluate the non-equilibrium flow [2, 3]. But a collision model in DSMC method plays an important role on the accuracy of the non-equilibrium flow. Tokumasu and Matsumoto presented the DMC (Dynamic Molecular Collision) model to simulate such a flow, and they showed the agreement between the numerical and the experimental results for one-dimensional shock wave[4]. The authors also simulated a flow around a flat plate with a sharp leading edge by using the collision model [5, 6, 7].

The purpose of the present research is to estimate the effect of non-equilibrium, the non-uniform flow and the boundary layer from the nozzle around a object by using two-dimensional cylindrical DSMC method from the viewpoint of engineering application.

NUMERICAL METHOD

The present DSMC method adopts the DMC model for gas-gas collision model and the diffuse model for gas-surface interaction. The DMC model of diatomic molecules is based on the collision dynamics, and this is featured by the cross sections and energy distributions after molecular collisions which are obtained by the MD simulations. For collision-frequency calculation, the null-collision technique [8] is adopted in the DSMC simulation. Downstream conditions in

CP762, *Rarefied Gas Dynamics: 24th International Symposium,* edited by M. Capitelli

Report Documentation Page	*Form Approved* *OMB No. 0704-0188*

1. REPORT DATE **13 JUL 2005**	2. REPORT TYPE **N/A**	3. DATES COVERED **-**

4. TITLE AND SUBTITLE **DSMC Simulation of Non-uniform Flow Effects behind a Conical Nozzle**	5a. CONTRACT NUMBER 5b. GRANT NUMBER 5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)	5d. PROJECT NUMBER 5e. TASK NUMBER 5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **Institute of Space and Astronautical Science, Japan Aerospace Exploration Agency, 3-1-1, Yoshinodai, Sagamihara, Kanagawa 229-8510, Japan**	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S) 11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT

Approved for public release, distribution unlimited

13. SUPPLEMENTARY NOTES

See also ADM001792, International Symposium on Rarefied Gas Dynamics (24th) Held in Monopoli (Bari), Italy on 10-16 July 2004. , The original document contains color images.

14. ABSTRACT

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			
unclassified	**unclassified**	**unclassified**	**UU**	**6**	

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std Z39-18

the present simulations are vacuum condition.

The nozzle in the simulation is a divergent conical nozzle. Because the computational cost reduces, the computational domain begins from the nozzle throat. The throat diameter of the nozzle is 13.7mm, the nozzle exit diameter is 100 mm, and nozzle exit half angle is 15 degrees, respectively. Therefore expansion ratio is equal to 53.18, and the exit Mach number for inviscid flow is six. The object is located at X =15mm downstream of the nozzle exit. Its diameter is 5 mm because it has to be located inside the core flow. The half angles of the objects are 10, 30, and 45 degrees.

Simulation conditions are listed in Table 1. Upstream values are also shown in it. Kn and Re numbers are calculated by the values in the core flow at the nozzle exit. No.1∼4 are the cases for the simulations with the nozzle. No.5∼7 are the cases for those without the nozzle, and upstream conditions are assume to be an uniform flow. Computational cells in the simulations are 6054 for No.1∼4 and 1824 cells for No.5∼7, respectively. Examples for the computational cells are shown in Figs. 1 and 3. The number of particles in the present simulations is approximately 200,000.

TABLE 1. DSMC simualtion conditions.

No.	1	2	3	4	5	6	7
Nozzle	yes	←	←	←	no	←	←
Half angle of object θ(deg.)	-	10	30	45	10	30	45
Upstream temperature T_∞ (K)	557	←	←	←	123.5	←	←
Upstream pressure p_∞ (Pa)	519	←	←	←	2.235	←	←
Upstream Mach number M_∞	1.0	←	←	←	4.67	←	←
Kn_e based on 0.05m	0.020	←	←	←	0.020	←	←
Re_e based on 0.05m	370	←	←	←	370	←	←
Wall temperature T_w(K)	300	←	←	←	←	←	←

RESULTS AND DISCUSSIONS

Effects of objects behind the nozzle

Figure 1 shows Mach number contours around the objects. Thickness of the boundary layer is approximately half of the nozzle exit diameter at the nozzle exit. A shock wave is formed in the vicinity of the nose of the object, and the shock wave and a boundary layer from the nozzle interact. Figure 2 shows translational and rotational temperature profiles over and on the object. The location at the nozzle exit and the nose of the object are X=189.4mm, 204.0mm, respectively. The difference between these temperatures in the core flow is approximately 10K as shown in the results for No.1, and small non-equilibrium due to rotational frozen appears in the nozzle. The diameter of the core flow is 30mm at the nozzle exit. For the case of No.2, non-equilibrium is observed near the object surface, and the maximum difference of both temperatures is 200K at X=210 mm. The results for No.3 and 4 differ from those for No.2 because the strong shock wave is generated in front of the nose.

Effects of the nozzle flow around the objects

Figure 3 shows the comparison between the results behind the nozzle and those in the uniform flow. Flow feature around the object is similar instead of the shock wave far from the object. Figures 4 ∼ 6 show the comparison for each object. It is shown that the results with the nozzle exit flow are larger than those for the uniform flow, and it is not dependent on the object shape. Translational and rotational temperatures for the uniform flow are approximately 20K lower than those with the nozzle exit flow. This would cause the non-equilibrium flow effect in the nozzle rather than the boundary layer developed along the nozzle wall and the flow gradient. Therefore it can be concluded that the simulation for the objects in the uniform flow based on data which are measured at the nozzle exit would overestimate temperature around them. The detailed mechanism on both temperatures due to the non-equilibrium effect is unclear, but we have to care the experimental data measured in the ground environment.

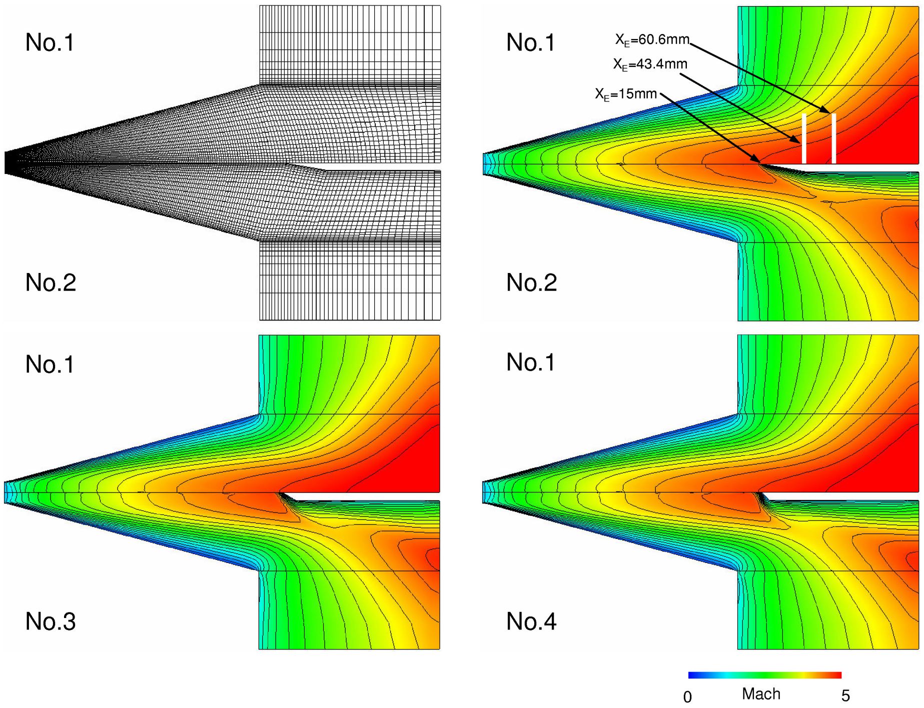

FIGURE 1. Computational grids and Mach contours for No.1 $\sim$ No.4

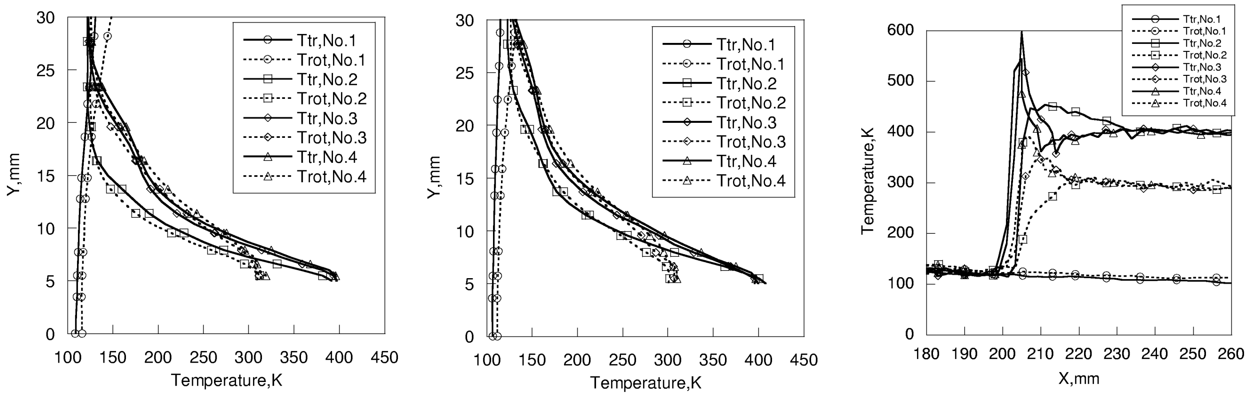

FIGURE 2. Temperature profiles around the object : left: X_E=43.4 mm(X=232.7mm); center: X_E=60.6 mm(X=250mm); right : on the object.

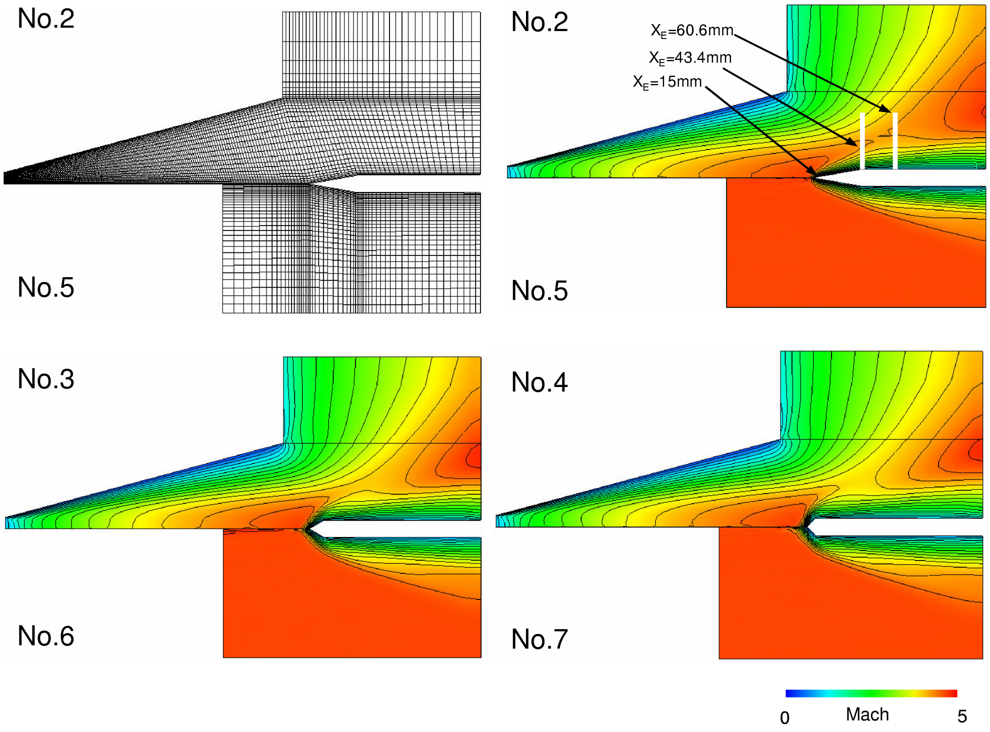

FIGURE 3. Computational grids and Mach contours for No.2 ∼ No.7

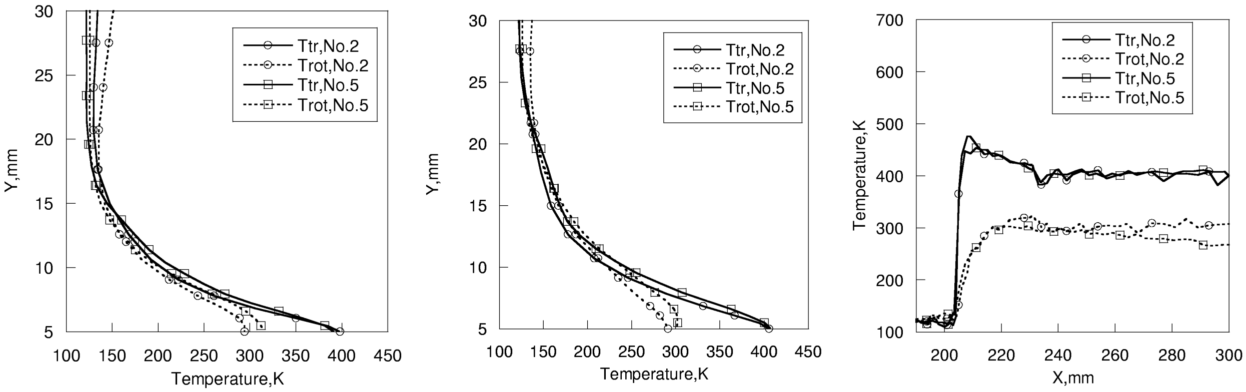

FIGURE 4. Temperature profiles around the object with θ=10 deg.: left: X_E=43.4 mm(X=232.7mm); center: X_E=60.6 mm(X=250mm); right : on the object.

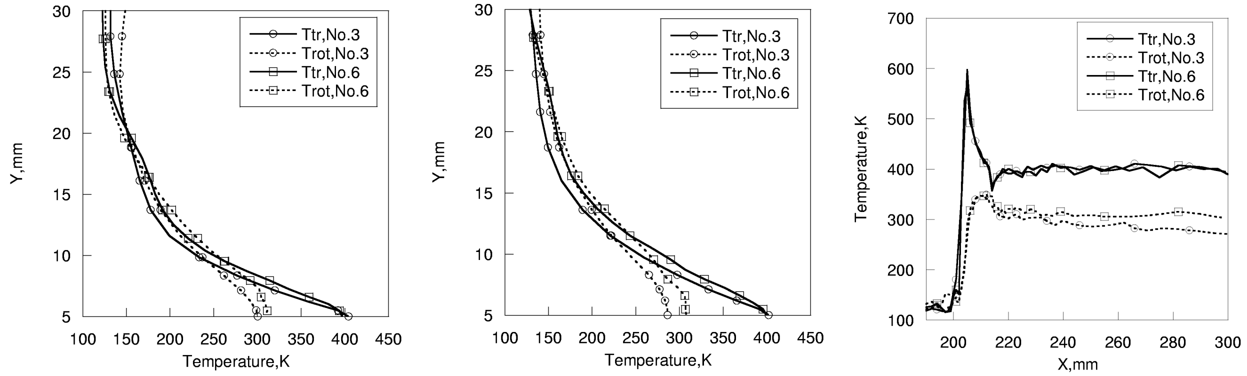

FIGURE 5. Temperature profiles around the object with θ=30 deg.: left: X_E=43.4 mm(X=232.7mm); center: X_E=60.6 mm(X=250mm); right : on the object.

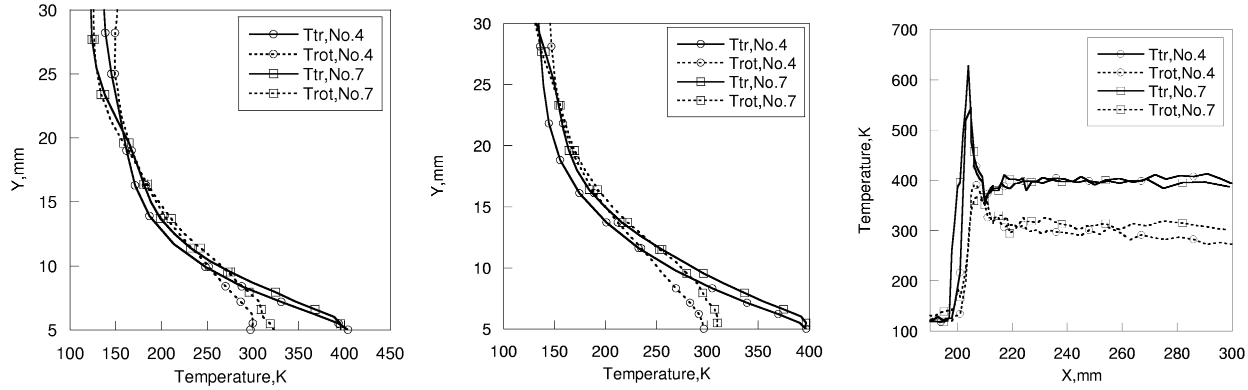

FIGURE 6. Temperature profiles around the object with θ=45 deg.: left: X_E=43.4 mm(X=232.7mm); center X_E=60.6 mm(X=250mm); right : on the object.

CONCLUSIONS

The two-dimensional cylindrical DSMC simulations around a object are conducted in order to estimate the effects of the upstream conditions from the viewpoint of engineering application. Non-equilibrium between translational and rotational temperatures at the nozzle exit is approximately 10K, and the diameter of the core flow is 30mm. The shock wave interacts the boundary layer developed along the nozzle wall significantly. The temperature profiles around the object are influenced by the upstream flow conditions, and that the difference is approximately 20K. The factors caused the discrepancy are thought to be non-equilibrium, boundary layer developed along the nozzle wall, and non-uniform flow. Therefore these effects should be estimated when the experimental results by using the nozzle flow compare with the numerical results.

REFERENCES

1. Lengrand, J., Allégre, J., J.Chpoun, A., and Raffin, M., "Rarefied Hypersonic Flow over a Flat Plate: Numerical and Experimental Results," in *18th Int. Symp. on Rarefied Gas Dynamics*, edited by R. Shizgal and D. Weaver, 160, AIAA, 1992, pp. 276–284.
2. Bird, G., *Molecular Gas Dynamics*, Calrendon Press, Oxford, 1976.
3. Nanbu, K., *J. Phys. Soc. Jpn*, **49**, 2042–2049 (1988).
4. Tokumasu, T., and Matsumoto, Y., *Physics Fluids*, **11**, 1907–1920 (1999).
5. Tsuboi, N., and Matsumoto, Y., "Numerical Simulation of Interaction between Shock Waves and Boundary Layer in Nonequilibrium Hypersonic Rarefied Flow," in *Proceedings of the 22nd International Symposium on Shock Waves*, edited by G. Ball, R. Hillier, and G. Roberts, 2, University of Southampton, 1999, pp. 909–914.
6. Tsuboi, N., Yamaguchi, H., and Matsumoto, Y., *AIAA paper 2001-1851* (2001).
7. Tsuboi, N., Yamaguchi, H., and Matsumoto, Y., *Journal of Spacecrafts and Rockets*, **41**, 397–405 (2004).
8. Koura, K., *Physics Fluids*, **29**, 3509–3511 (1986).