Numerical Studies on Effect of Nozzle Geometry and of Flow Continuity on Thrust Performance

S.Hatta*, S.Aso†
Space Transportation Systemd Engineering Laboratory
Department of Aeronautics and Astronautics, Kyushu University
6-10-1 Hakozaki, Higashi-ku, Fukuoka-shi, 812-8581, Japan
81-092-642-4138
hatta@aero.kyushu-u.ac.jp

IEPC-01-193

Numerical studies have been conducted in order to investigate the effects of nozzle geometry and rarefaction on the thrust performance. For the estimation, ratio of mean free path to calculation grid size, grid Knudsen number is newly installed and evaluated. Based on the grid Knudsen number, wide range of nozzle flow is considered to be rarefied gas. Thus, continuum flow is solved by N-S equation and rarefied gas region is solved by DSMC. The results show that the rarefaction have some effect on velocity distribution.

1. Introduction

It has already been reported in many studies that so-called nozzle efficiency is around 85~95 percent. In order to increase nozzle efficiency, more precise flow study should be conducted [1-6]. However, those studies have not been conducted sufficiently.
In the previous study, numerical simulations are conducted, and effects of aperture of the nozzle and mass flow rate on thrust are reported [7].
In the present study, numerical simulations are applied for similar flow field of 2kW class arcjet-thruster.
First in the present study, numerical simulation is conducted for three different nozzle shapes of half angle of 10º, 20º and 30º. Thickness of boundary layer and its influence on thrust performance is considered in each case. Also influence of rarefaction on the flow is considered. To evaluate rarefaction of flow, ratio of mean free path to calculation grid size is used. From the evaluation, wide range of nozzle flow is considered to be rarefied gas.
The results show that the newly developed hybrid method is useful to obtain more precise velocity distribution at nozzle exit [8].

2. Numerical Method

2.1 Outline of Numerical Method

Three types of numerical methods are used in the present study. Those are
1) Navier-Stokes equation with physical inflow model,
2) N-S equation with Joule Heating,
3) The Direct Simulation Monte Carlo (DSMC).
Details of them are described below.

2.2 Numerical Analysis with N-S Equation

The fundamental axisymmetric N-s equations, which hold at chemical nonequilibrium and thermal equilibrium state, are as follows:

1) Equation of conservation of total mass
2) Equation of axial momentum
3) Equation of radial momentum
4) Equation of conservation of total energy

*Graduate student
†Professor

5) Equation of conservation of each species

The model of the flow field

The following assumption are applied for the flow field:

1) The propellant is nitrogen.
2) Continuous flow
3) Laminar flow
4) Axisymmetric flow
5) Ignoring the effect of electric field and magnetic field
6) Chemical nonequilibrium flow
7) 5 species (N^2, N, N_2^+, N^+, e^-)
8) One temperature model
9) The effect of radiation is neglected.

Chemical reaction model

5 species and 8 chemical elementary reactions are assumed in the present calculation.

(1) Heavy Particle-Impact Dissociation

$$N_2 + M \Leftrightarrow N + N + M \ (M = N_2, N, N_2^+, N^+)$$

(2) Electron-Impact Dissociation

$$N_2 + e^- \Leftrightarrow N + N + e^-$$

(3) Associative Ionization

$$N + N \Leftrightarrow N_2^+ + e^-$$

(4) Charge Exchange

$$N^2 + N^+ \Leftrightarrow N_2^+ + N$$

(5) Electron-Impact Ionization

$$N + e^- \Leftrightarrow N^+ + e^- + e^-$$

Nozzle shape and Computation Grid

The nozzle geometry is obtained from the experimental setup as shown in Fig.1, 2, 3.

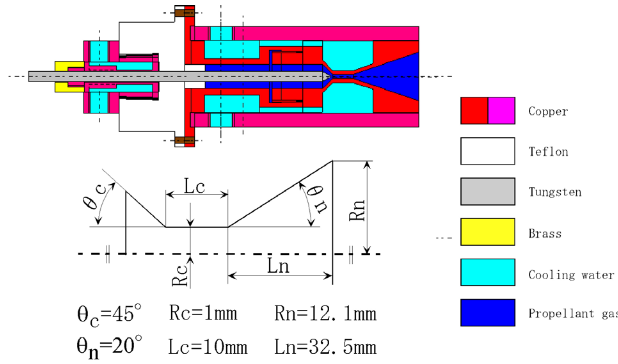

Fig.1 Schematic of Arcjet Thruster

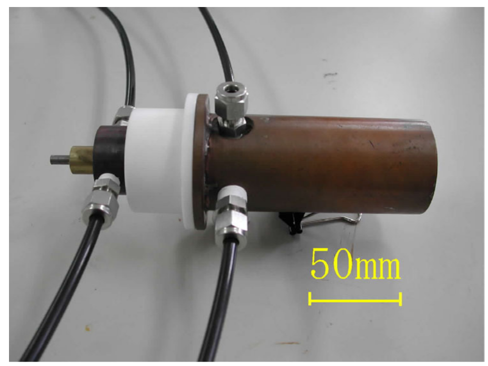

Fig.2 Arcjet Truster

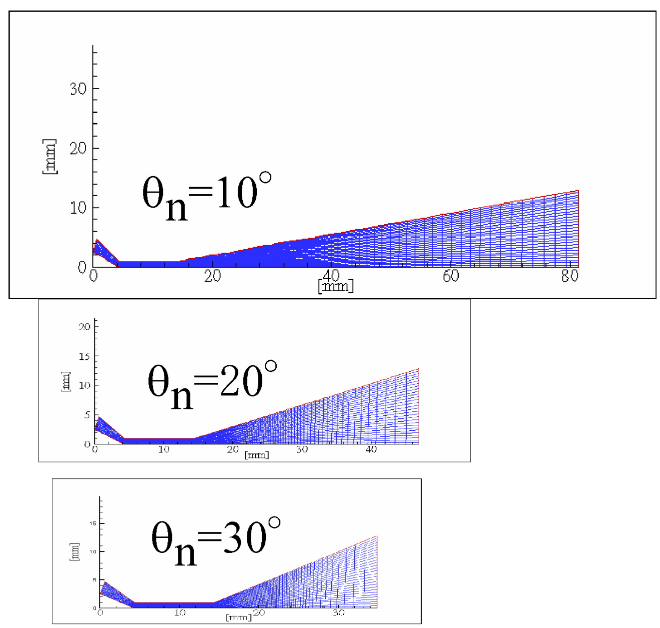

Fig.3 Computation Grid

Boundary conditions

The nozzle wall is assumed as adiabatic and non slip. At nozzle exit, outflow boundary condition with zero derivative with respect to main flow direction is used. It is since the flow is subsonic at the inlet, that one of the flow variables is extrapolated from downstream vicinity of inlet. In this case static pressure is used as the variable under condition of constant mss flow rate and of constant stagnation temperature T_0.

In the numerical simulation of N-S equation with physical inflow model, the gas temperature after arc heating is estimated by considering arc heating. The flow's physical model is assumed:

$$mc_pT_0 + \eta P = mc_pT_0^* \qquad (1)$$

where,

m :mass flow rate

c_p :specific heat at constant pressure

P :charging electric power

η :thermal efficiency.

In the model, propellant (stagnation temperature: T_0) is heated up to different stagnation temperature of T_0^*. T_0^* is used as the stagnation temperature of propellant in chapter 3.

Joule Heating

To investigate the effect of rarefaction in chapter 4, DSMC method is conducted. In order to impose the inflow condition for the DSMC calculation, N-S equation with Joule heating is also calculated.

The current density required for estimation of Joule heating is calculated from combination of the generalized ohm's low and the steady state Maxwell's equation,

$$\nabla \cdot (\sigma \nabla \phi) = \nabla \cdot \left(\frac{\sigma}{en_e} \nabla p_e \right) \qquad (2)$$

2.3 Numerical Analysis with the DSMC Method

In some prospective experimental cases, propellant is so rarefied that it is no longer sustain continuum. Therefore, the DSMC method is conducted to such rarefied flow field. The follow assumption for DSMC is obtained by estimating numerical result of N-S with Joule Heating as follows,

1) Fluid consists of N_2 and N.
2) Dissociation, ionization and recombination are ignored.
3) The Larsen-Borgnakke model with vibrational energy relaxation is adopted.
4) Diffuse reflection is applied at wall.

3. Nozzle Geometry Effect

As mentioned above, numerical analysis with physical inflow model of equation(1) is applied to flow field of three different nozzle geometries under conditions of Table.1

Table.1

(a) Mass Flow Rate of 10 slm

Half Cone Angle	10°	20°	30°
P [W]	2000	2000	2000
Total Temperature [K]	2687.2	2687.2	2687.2
Re	2020	1450	1010
Isp	220	225	222

(b) Mass Flow Rate of 100 slm

P [W]	20000	20000	20000
Total Temperature [K]	2687.2	2687.2	2687
Re	10200	8820	6910
Isp	240	236	224

Mass flow rates are 10 slm and 100 slm. Input electric power are 2000W and 20000W respectively in order to fix the total temperature to 2687.2 [K] under condition of $\eta = 0.3$.

Representative results of Isp are also shown in Table.1. The nozzle geometry with any half cone angle shows the increase of specific impulse along with the increase of mass flow rate for some half cone angle. This is considered as follows. *Re* number is increased by increase of mss flow rate. The increased *Re* number make boundary layer thin which result in the increase of velocity near nozzle wall as shown in Fig.4a, 4b, 4c.

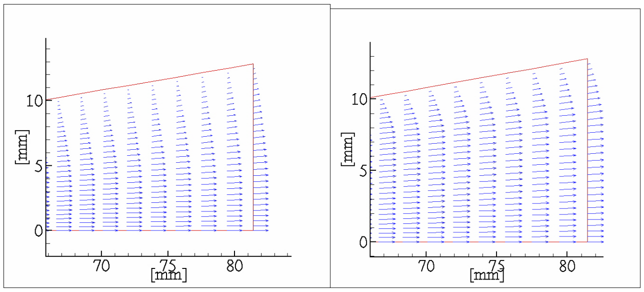

Fig.4a Half Cone Angle of 10º

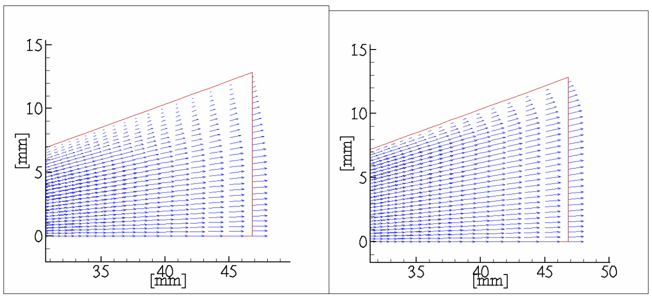

Fig.4b Half Cone Angle of 20º

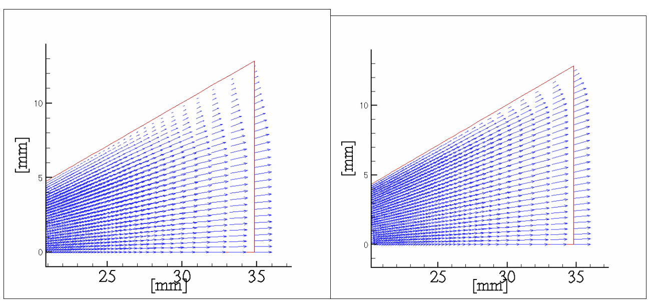

Fig.4c Half Cone Angle of 30º

However, increasing amount of Isp is the largest at the geometry with half cone angle of 10º. Because the boundary layer develops well on the longest nozzle wall at half cone angle of 10º and the boundary layer thickness is influenced by *Re* number. At high mass flow rate, which decrease the influence of boundary layer, the nozzle geometry with small half cone angle shows the highest Isp. Also at half cone angle of 10º, axial component of momentum is largest. This can be observed from vector distribution of the result. Though, half cone angle of about 20º of arcjet thruster is believed most effective, the present result show much smaller half cone angle of about 10º is more useful to obtain higher Isp.

4. Effect of Rarefaction

The numerical simulation described above is effective under assumption of continuum of gas. In order to validate the assumption of continuum, the Knudsen number *Kn* is generally used.

$$Kn = \lambda / L \tag{3}$$

Where, λ is *mean free path* and L is the characteristic length of the flow.

In numerical simulation by solving N-S flow field are divided into small computational grid. In some critical conditions the local Knudsen number based on the characteristic length of grid size is quite larger than that based on the ordinary characteristic length. Even in those situation conservation lows of mss, momentum, and energy are usually expressed by using continuum fluid. However, in those critical situations considerably small number of molecular particles might exist in a grid cell and numerical solver based on N-S is not available. The present authors solve the follow field at first and estimate local Knudsen number;

$$Kn = \lambda / \Delta x \tag{4}$$

And the flow region, whose local Knudsen number is above 0.1 or 1, is solved by DSMC.

The representative result is shown below in Fig.5a, 5b.

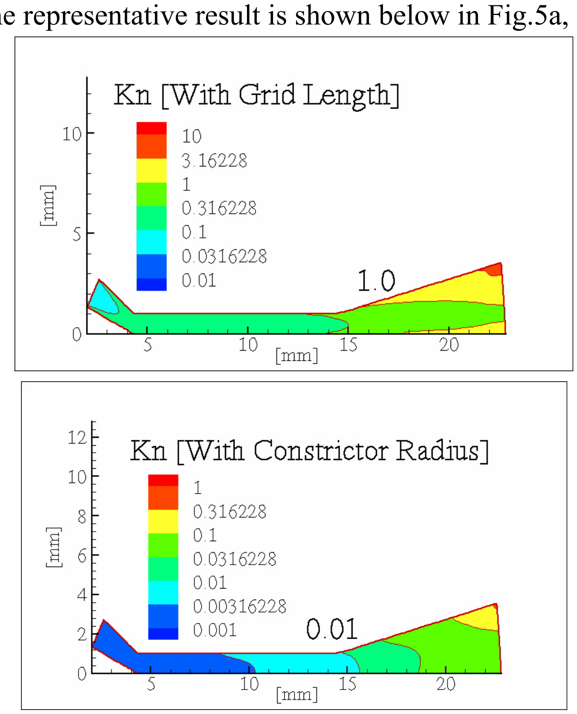

Fig.5a Analysis with The Physical Inflow Model

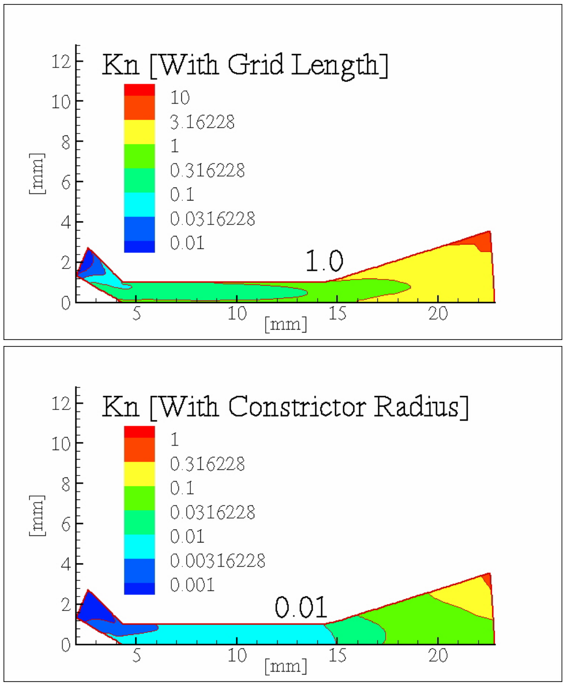

Fig.5b Analysis with Joule Heating

The mass flow rate is set at 41.68[mg/s](2 standard litter per minutes). In addition, as is shown obviously, the computational grid is restricted so that it can acquire more precise result as well as it is useless to calculate down to the rarefied region

In any cases, large *Kn* region covers by far the upstream of the nozzle. Hence, the flow field in arjet nozzle under such condition should be treated as a molecular flow and the DSMC method is conducted as mentioned above.

An example of result is shown in Fig.6 and the momentum component of thrust of specific impulse is shown in Table.2.

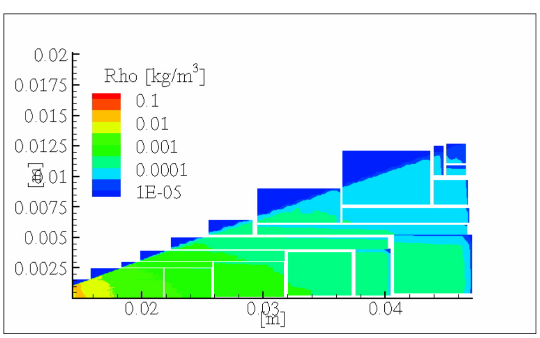

Fig.6 Distribution of Density Calculated by DSMC

Table.2 Momentum Thrust Element of Isp.

	N-S	DSMC
With Physical Flow Model	170 [s]	177 [s]
With Joule Heating	243 [s]	247 [s]

It is important that Isp are estimated at the downstream of about 8 mm from the exit of the constrictor in the cases of N-S, while about 32 mm in the cases of DSMC. However, only a little difference of thrust performance can be seen among them. Hence, it seems that the nozzle doesn't work as a supersonic nozzle downstream of the distance of 8 millimeter or a little far from the constrictor. It might be due to the influence of rarefaction, or the breakdown of the continuum gas assumption.

5 Summary

Numerical analysis with three different approaches has been attempted. The influence of half cone angle on arcjet nozzle flow field at high mss flow rate is shown and availability of the small half cone angle nozzle is suggested.

Local Knudsen number based on the grid size is introduced and numerical simulation by DSMC is also conducted for rarefaction case. The results show the local observation of rarefaction of gas flow is important.

Numerical analysis with the DSMC method is inducted and shows the influence of rarefaction.

Remains to be done

What remains to be done are,

1) Expansion of chemical species to simulate more practical propellant such as N_2H_4, NH_3, and so on.

2) Induction of the multi temperature model which is expected to simulate the flow field more accurately.

3) More realistic boundary conditions

4) More available DSMC code for various conditions of analysis.

In addition to these, experimental data is necessary to reveal the real aspect of the influence of nozzle geometry and rarefaction.

References

[1] John R.Brophy , Gary L.Bennett , Francis M.Curran, ”Overview of NASA’s Electric Propulsion Program”, IEPC Paper 95-1,1995

[2] Robert G.Jahn, ”Physics of Electric Propulsion” , McGraw-Hill,1968

[3] Shigeru Kuchi-ishi, ”Numerical Studies of Thermochemical Nonequilibrium Flows in a DC Arcjet Thruster” , Doctoral.thesis in Kyushu University ,1999

[4]K.Ogiwara,T.Yamada,K.Toki,K.Kuriki, ”Experiment of 300W-class Arcjet Thruster ”, ISTS 94-a-50,1994

[5]M.Riehle,T.Laux,F.Hauber,H.L.Kurtz,M.Auweter-Kurtz, ”Performance Evaluation of Regeneratively Cooled 1,10 & 100kW Arcjets”, IEPC-99-027,1999

[6]I.Funaki,K.Toki,Y.Shimizu,Y.Suwama,H.Takegahara,S.Satori ”Preliminary Test of a Law-Power DC Arcjet Anode”,IEPC-99-046,1999

[7]S.Aso,et al “A Study on Nozzle Flow Quality of Low Power Arcjet Thruster for Improvement of Propulsion Efficiency”,IEPC-99-036,1999

[8] G.A.Bird, ”Molecular Gas Dynamics and the Direct Simulation of Gas Flows” ,Oxford Science Publications,1995