Chinese Journal of Aeronautics, 2013,**26**(3): 522–528

Chinese Society of Aeronautics and Astronautics
& Beihang University
Chinese Journal of Aeronautics
cja@buaa.edu.cn
www.sciencedirect.com

CSAA

CHINESE JOURNAL OF AERONAUTICS

Gaseous plume flows in space propulsion

Wang Limei, Cai Chunpei *

Department of Mechanical and Aerospace Engineering, New Mexico State University, Las Cruces, NM 88003, USA

Received 4 February 2012; revised 30 March 2012; accepted 8 May 2012
Available online 30 April 2013

KEYWORDS

Free molecular flow;
Gasdynamics;
Gaskinetics jets;
Monte Carlo methods;
Plume flow

Abstract This paper presents a gaskinetic study on high-speed, highly rarefied jets expanding into a vacuum from a cluster of planar or annular exits. Based on the corresponding exact expressions for a planar or annular jet, it is convenient to derive the combined multiple jet flowfield solutions of density and velocity components. For the combined temperature and pressure solutions, extra attention is needed. Several direct simulation Monte Carlo simulation results are provided to validate these analytical solutions. The analytical and numerical solutions are essentially identical for these high Knudsen number jet flows.

1. Introduction

Jet and jet impingement flows are two fundamental fluid dynamics problems with numerous applications in multiple disciplines including physics, chemistry astronomy, and engineering. As an important limiting situation, the high Knudsen number *Kn* gaseous flow scenario has numerous applications as well, for example, molecular beams,[1,2] thermal/chemical/ion/plasma rocket plumes,[3,4] thin film deposition process inside a vacuum chamber,[5] and gaseous spray. The bi-annual international Rarefied GasDynamics (RGD) symposiums collect papers on rarefied jet, jet impingement and molecular beams. Collisionless flow solutions also provide new insights to investigate many gaseous flow problems.

For the propulsion community, gaseous jets or plumes play an important role, with applications of infrared radar detection and performance evaluations. Further, plume impingement has been a serious issue for spacecraft/aircraft design, and the plume impingement force, torque and heat fluxes must be computed to avoid control problems and damage to sensitive surfaces.

To study plume flows, there are several approaches, experiments, simulations, modeling and theoretical work. In the literature, there are numerous reports of all of these three approaches. For example, the direct simulation Monte Carlo method (DSMC)[6] is an effective approach to simulate rarefied jet and jet impingement flows. National Aeronautics and Space Administration (NASA) has spent an significant amount of effort investigating the plume and plume impingement effects on spacecraft. One major outcome is the versatile DSMC analysis code (DAC) package.[7]

As for the modeling and analytical approaches, plume models can be characterized as either detailed or analytical.[8] The former is based on theoretical treatments while analytical models are simpler representations based on more approximate treatments of the complex gas dynamic and radioactive processes.[8] One practical and popular formula is the Cosine law or the Boynton-Simons'plume model,[9] which has been widely used for over half a century. This model predicts only flowfield density distributions far from the rocket exit. Rocket nozzle flow may be treated as though it originates from a point source. Using the continuity equation with constant mass flux

* Corresponding author. Tel.: +1 575 646 7704.
E-mail addresses: wlm@nmsu.edu (L. Wang), ccai@nmsu.edu (C. Cai).
Peer review under responsibility of Editorial Committee of CJA.

ELSEVIER Production and hosting by Elsevier

1000-9361
http://dx.doi.org/10.1016/j.cja.2013.04.049

across different spherical surfaces, the density at any point in the plume can be computed in a closed form.

Another important established category of simplified models for high speed jets or plumes are highly rarefied flows out of a nozzle with a nonzero average velocity. Narasimha[10] illustrated that the density solutions for collisionless jet flow from a circular exit are much more complex than the Cosine law. Noller[11] adopted a solid angle approach to compute the local plume density field from an exit. By using discrete point sources, Brook predicted the density flowfield from an annular leakage in a mis-sealed spacecraft hatch.[12] Woronowicz and Ghaffarian[13] proposed a treatment of a starting surface, based on which they divided the highly rarefied plume fields as a collection of collisionless point sources, as such, density and velocity fields can be obtained conveniently. In the previous studies, Cai and Boyd[14,15] established an approach and applied it to obtain two sets of detailed solutions for collision-less planar and circular plume flows. The solutions are further extended to the problems of jet impingement[16,17] at a normally set flat plate.

A recent continuing development on rarefied, plume flows compares the new set of analytical collisionless plume flow solutions[18] and the Cosine law or the Boynton–Simons' plume model. These major conclusions found are:

(1) The gaskinetic solutions compute the density results over the whole flowfield accurately; the Cosine law or the Boynton–Simons' plume model has poorer performance in near field due to the point source assumption.
(2) Other than the density field, the new gaskinetic solutions provide extra information of velocity, temperature and pressure results than the Cosine law or the Boynton–Simons' plume model.
(3) The gaskinetic solutions are developed for collisionless jet or plume flows. They approximate high speed gaseous flows well, even at a near continuum regime. With a high flow speed, gas molecules have less time to collide and diffuse from the main flow direction.

A cluster of gaseous jets or plumes have important applications as well. Examples include: (A) an array or a matrix of gaseous sprays for combustion; (B) inside a vacuum chamber for materials processing, multiple nozzles inject different metal powders. For high speed jets into vacuum, distributions at the exit need to consider multiple jets of different strength and a cluster of thermal/chemical/ion/plasma rocket plumes.

The last application is of special interest to the propulsion and power community. For example, Hall thrusters represent a very efficient form of electric propulsion devices widely used on spacecraft for primary propulsion and on-orbit applications such as station keeping. A cluster of thrusters is more favorable than a larger thruster with several advantages: lower manufacturing cost, less demanding requirement on test facilities, and more robustness and capability to tolerate failure of single thruster. Numerical simulation models of plasma flows from a cluster of Hall thrusters have been developed to aid our studies on spacecraft integration issues.[19] There are methods to directly simulate three-dimensional plasmas flows from a cluster of two or four Hall thrusters. However, simulations of three-dimensional models are time-consuming and inconvenient, and it is desirable to develop a set of accurate, efficient, analytical gaskinetic solutions.

This paper demonstrates a procedure to compute the combined flowfield properties from multiple jets, where the pressure field computation is not trivial. We adopt two planar and annular jets to demonstrate the flowfield solutions. In the same vein, the results can be conveniently extended to an array or a matrix of jets.

2. A collisionless jet from a planar or annular exit

For a high-speed gaseous jet or plume flow from a planar or annular exit, it is common and reasonable to assume that gas at the exit is in an equilibrium state which is characterized by a Maxwellian distribution with a number density, n_0, an exit temperature, T_0, and an average macroscopic velocity, U_0. An approach[20] has been established and further developed to obtain the exact analytical solutions for high Kn number jet and plume flows. The crucial point of the approach is to determine a right group of molecules which are from the exit and arrive at a specific flowfield point. These molecules' thermal velocities shall form a special velocity domain; without any collisions, this velocity domain must be the same one for a flowfield point. The velocity domain at the exit is regulated with a crucial fundamental relation of velocity direction and geometry location: for those particles from a nozzle exit point $(0, y, z)$ and reaching a flowfield point (X, Y, Z), their thermal velocity components shall follow a fundamental constrain relations[14,15]:

$$\frac{X-0}{u+U_0}=\frac{Y-y}{v}=\frac{Z-z}{w} \tag{1}$$

where u, v, w are the thermal velocity.

This reveals that fundamental relation leads to the solutions of density, velocity components, and temperature and pressure fields.[15,20]

2.1. Single planar jet

The flowfield results for a single free planar jet into vacuum are[15,16]:

$$\begin{aligned}\frac{n_1(X,Y)}{n_0}=&\frac{\exp(-S_0^2)}{2\pi}(\theta_2-\theta_1)+\frac{1}{4}[\mathrm{erf}(S_0\sin\theta_2)\\&-\mathrm{erf}(S_0\sin\theta_1)]+\frac{S_0}{2\sqrt{\pi}}\int_{\theta_1}^{\theta_2}\exp(-S_0^2\sin^2\theta)\\&\times\cos\theta\,\mathrm{erf}(S_0\cos\theta)\mathrm{d}\theta\end{aligned} \tag{2}$$

where $S_0=U_0\sqrt{\beta_0}$ is the exit speed ratio, β_0 is the characteristic thermal velocity, θ is the position angle, $\theta_1=\mathrm{atan}((Y-H)/X)$ and $\theta_2=\mathrm{atan}((Y+H)/X)$ are special geometry angles,[14,20] $\mathrm{atan}(\cdot)$ is the inverse tangent function, H is the semi-height or width for a single nozzle, $\mathrm{erf}(\cdot)$ is the error function, subscript "1" represents free plume results. The macroscopic velocity components U_1, V_1 and the temperature T_1, are as follows:

$$\begin{aligned}\sqrt{\beta_0}U_1(X,Y)=&\frac{n_0\exp(-S_0^2)}{2n_1\pi}\\&\cdot\left[\frac{1}{2}(\theta_2-\theta_1)+\frac{1}{4}(\mathrm{erf}(S_0\sin\theta_2)-\mathrm{erf}(S_0\sin\theta_1))\right.\\&\left.+\frac{S_0}{2\sqrt{\pi}}\int_{\theta_1}^{\theta_2}\exp(-S_0^2\sin^2\theta)\cos\theta\,\mathrm{erf}(S_0\cos\theta)\mathrm{d}\theta\right]\end{aligned} \tag{3}$$

$$\sqrt{\beta_0}V_1(X,Y)=\frac{n_0}{4n_1\sqrt{\pi}}\cdot\left[\exp(-S_0^2\sin^2\theta_1)\cos\theta_1 A(\theta_1)-\exp(-S_0^2\sin^2\theta_2)\cos\theta_2 A(\theta_2)\right] \quad (4)$$

where

$$A(x) = 1 + \mathrm{erf}(S_0\cos(x))$$

$$\frac{T_1(X,Y)}{T_0} = -\frac{U_1^2+V_1^2}{3RT_0}+\frac{n_0\exp(-S_0^2)}{6n_1\pi}\cdot\left[(3+S_0^2)(\theta_2-\theta_1)+\frac{1}{2}S_0^2(\sin(2\theta_2)-\sin(2\theta_1))+2\sqrt{\pi}\int_{\theta_1}^{\theta_2}(2+S_0^2\cos^2\theta)S_0\cos\theta\exp(S_0^2\cos^2\theta)A(\theta)\mathrm{d}\theta\right] \quad (5)$$

where R is the Argon gas constant, and the pressure result is computed with the equation of state.

2.2. Single annular jet

The results for a single annular jet are as follows:

$$\frac{n_1(X,0,Z)}{n_0} = \frac{\exp(-S_0^2)}{\sqrt{\pi^3}X^2}\int_{-\pi/2}^{\pi/2}\mathrm{d}\varepsilon\int_{R_1}^{R_2} rK\mathrm{d}r \quad (6)$$

where ε, r are the integration variables for the polar coordinate system; and the velocity components U_1 and W_1 along the flow and perpendicular flow directions, and the temperature fields T_1 are:

$$U_1(X,0,Z)\sqrt{\beta_0} = \frac{\exp(-S_0^2)}{\sqrt{\pi^3}X^2}\frac{n_0}{n_1}\int_{-\pi/2}^{\pi/2}\mathrm{d}\varepsilon\int_{R_1}^{R_2} rM\mathrm{d}r \quad (7)$$

$$W_1(X,0,Z)\sqrt{\beta_0} = \frac{\exp(-S_0^2)}{\sqrt{\pi^3}X^3}\frac{n_0}{n_1}\int_{-\pi/2}^{\pi/2}\mathrm{d}\varepsilon\int_{R_1}^{R_2}(Z-r\sin\theta)rM\mathrm{d}r \quad (8)$$

$$\frac{T_1(X,0,Z)}{T_0} = -\frac{U_1^2+W_1^2}{3RT_0}+\frac{4}{3}\frac{\exp(-S_0^2)}{\sqrt{\pi^3}X^2}\frac{n_0}{n_1}\times\int_{-\pi/2}^{\pi/2}\mathrm{d}\varepsilon\int_{R_1}^{R_2}N r\mathrm{d}r \quad (9)$$

Here the integration factors Q, K, M, N are as follows:

$$Q = \cos^2\psi\left[\sum_{n=0}^{\infty}P_n(\sin\psi\sin\varepsilon)\left(\frac{r}{\sqrt{X^2+Z^2}}\right)^n\right]^2 \quad (10)$$

where $P_n(\cdot)$ denotes the Legendre polynomials, and ψ = arctan(Z/X).

$$K = Q\left[QS_0+\left(\frac{1}{2}+QS_0^2\right)\sqrt{\pi Q}\left(1+\mathrm{erf}\left(S_0\sqrt{Q}\right)\right)\exp(S_0^2Q)\right] \quad (11)$$

$$M = Q^2\left[QS_0^2+1+S_0\left(\frac{3}{2}+QS_0^2\right)\sqrt{\pi Q}\left(1+\mathrm{erf}\left(S_0\sqrt{Q}\right)\right)\exp(S_0^2Q)\right] \quad (12)$$

$$N = S_0Q^2\left(\frac{5}{4}+\frac{QS_0^2}{2}\right)+\frac{1}{2}\sqrt{Q^3\pi}\left(\frac{3}{4}+3QS_0^2+Q^2S_0^4\right)\cdot\left(1+\mathrm{erf}\left(S_0\sqrt{Q}\right)\right)\exp(S_0^2Q) \quad (13)$$

These single planar or annular thruster solutions are crucial because they are applicable for rarefied or high speed continuum flows.[18] They are also the foundations for multiple jets.

3. Rarefied planar or annular jets into vacuum

3.1. Multiple planar jets into vacuum

This section utilizes the above results to obtain the multiple planar jet solutions. The results can be extended to an array of planar jets, or to compute a nozzle exit with non-uniform properties of n_0, T_0, and U_0. Fig. 1 depicts two planar jets, where D is the distance between two thruster centers. The corresponding thermal velocity phase for a specific flowfield point $A(X, Y)$ is illustrated in Fig. 2 with two separate regions. The region within θ_3 and θ_4 represents the contribution from the upper jet, while the other within θ_2 and θ_1 represents the contribution from the lower jet.

It can be proven that the combined flowfield from the cluster relates to the single jet properties. We assume that the two jets are not identical, i.e., the number density n_0 and temperature T_0 at the exit are different, then

$$n_{12} = n_1(X,Y)+n_2(X,Y) \quad (14)$$

$$n_{12}(X,Y)U_{12}(X,Y) = n_1(X,Y)U_1(X,Y)+n_2(X,Y)U_2(X,Y) \quad (15)$$

$$n_{12}(X,Y)V_{12}(X,Y) = n_1(X,Y)V_1(X,Y)+n_2(X,Y)V_2(X,Y) \quad (16)$$

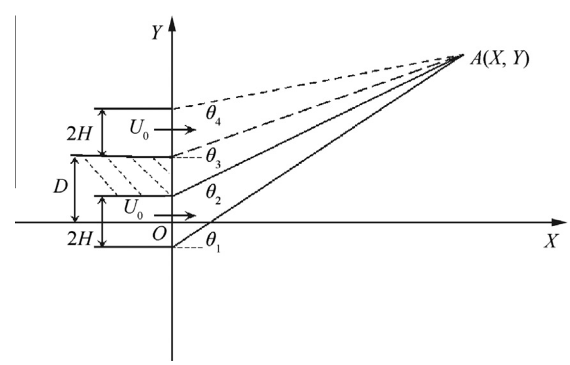

Fig. 1 Planar jets: problem illustration.

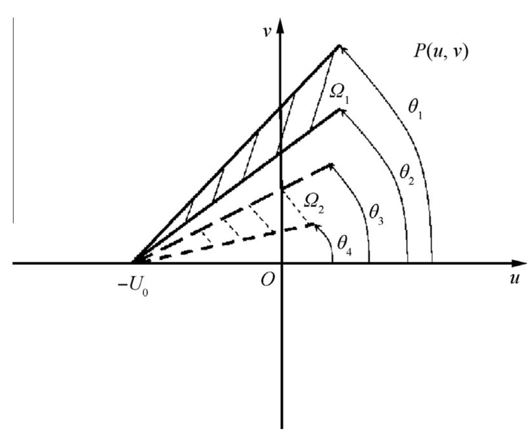

Fig. 2 Planar jets: velocity phase for one flowfield point.

The combined pressure fields can be derived from the Dalton's partial pressure law. Bearing in mind that the partial pressure is based on the combined global flowfield velocities, not on each single jet separately:

$$P_{12} = p_1 + p_2 + \frac{m}{3}\Big[n_1(U_1 - U_{12})^2 + n_1(V_1 - V_{12})^2 + n_2(U_2 - U_{12})^2 + n_2(V_2 - V_{12})^2\Big] \quad (17)$$

where P_{12}, U_{12} and V_{12} are the combined global macroscopic pressure and velocity components of two jets, P_1, U_1, V_1 and P_2, U_2, V_2 the corresponding single jet properties. The previous results of single jet, Eqs. (2)–(5), actually provide one approach to compute the exact combined flowfield for multiple planar jets. One more complicated scenario is that the exit speed U_0 is different for each exit. For that situation, the phase velocity domains Ω_1 and Ω_2 do not share the same vertex point.

3.2. Multiple annular jets into vacuum

Another important application of the previous analytical results is a cluster of annular jets, such as those from a cluster of Hall thrusters. The problem can be decoupled with two local coordinates illustrated by Fig. 3, and the thermal velocity phase domains by Fig. 4. At a point $A(X, Y, Z)$ in the flowfield, the velocity phase domains consist of two separate regions; each actually contains two hollow cones sharing the same vertex. The flowfield of two annular jets can be computed locally with those single jet results correspondingly. These contributions are then transformed and added into a global coordinate system.

The procedure is similar to that for the planar case. Because the results for a cluster of annular exits are actually three-dimensional, the process is more complex. Different Q_1 and Q_2 shall be carefully defined, and the integration shall consider the third velocity component, W, for the pressure and temperature fields. In the same vein as the planar scenario, the results for the number density, U- and V-velocity components are similar to Eqs. (14)–(16). Furthermore,

$$n_{12}(X, Y, Z)W_{12}(X, Y, Z) = n_1 W_1(X, Y, Z) + n_2 W_2(X, Y, Z) \quad (18)$$

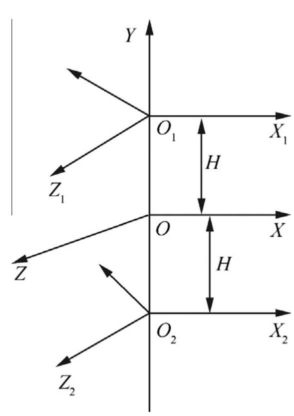

Fig. 3 Annular jets: coordinate definitions.

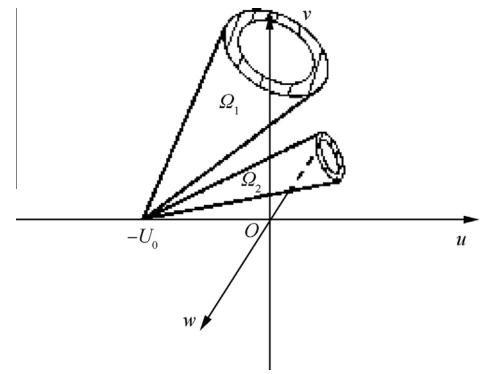

Fig. 4 Annular jets: velocity phase for one flowfield point.

$$P_{12} = p_1 + p_2 + \frac{m}{3}\Big[n_1(U_1 - U_{12}^2 + n_1(V_1 - V_{12})^2 + n_1(W_1 - W_{12})^2 + n_2(U_2 - U_{12})^2 + n_2(V_2 - V_{12})^2 + n_2(W_2 - W_{12})^2\Big] \quad (19)$$

where W_{12} is the macroscopic velocity component along the Z-direction for the combined global flowfield of the two jets; W_1 and W_2 are the corresponding single jet results.

The above results and approach are not as trivial as they look, and they can provide us with more insights. During the past, without the information of exact collisionless pressure field solutions, different assumptions are proposed. For example, it is proposed to compute the plume pressure field with local plume density results times the exit temperature T_0. Another common practice is due to the fact that the gas is highly rarefied, and separate pressure fields are added up to obtain the combined pressure field, such as a jet plume flow inside a vacuum chamber with background pressure; and a starter surface treatment for plumes. Eqs. (17) and (19) illustrate that these approaches are incorrect.

4. Validations

This section provides some DSMC simulation results to validate the above results. For the case of a single planar or an annular jet, the formulae have been verified.[18] As such, this section only includes some validation work for the solutions to a cluster of planar or annular jets with the same exit conditions. All simulations have $Kn = 100$, based on the exit width and that the test gas is argon. For the two planar jet cases, the exit width is 0.1 m and the jet center-to-center distance is 0.5 m. For the annular case, the nozzle inner and outer radii are set to $R_2 = 0.0150$ m, $R_1 = 0.0075$ m, and the center-to-center distance is 0.5 m. This is a special setup of two annular thrusters.[20] A specific DSMC package which we used, GRASP-P,[21] is a general purpose particle simulation package developed with special object-oriented programming styles and design patterns.

Fig. 5 compares theoretical and numerical results of density, velocity components and pressure for a cluster of two planar jets with the same exit conditions. Because the problem is symmetric, the simulation is performed with one thruster by adopting a symmetric boundary condition.

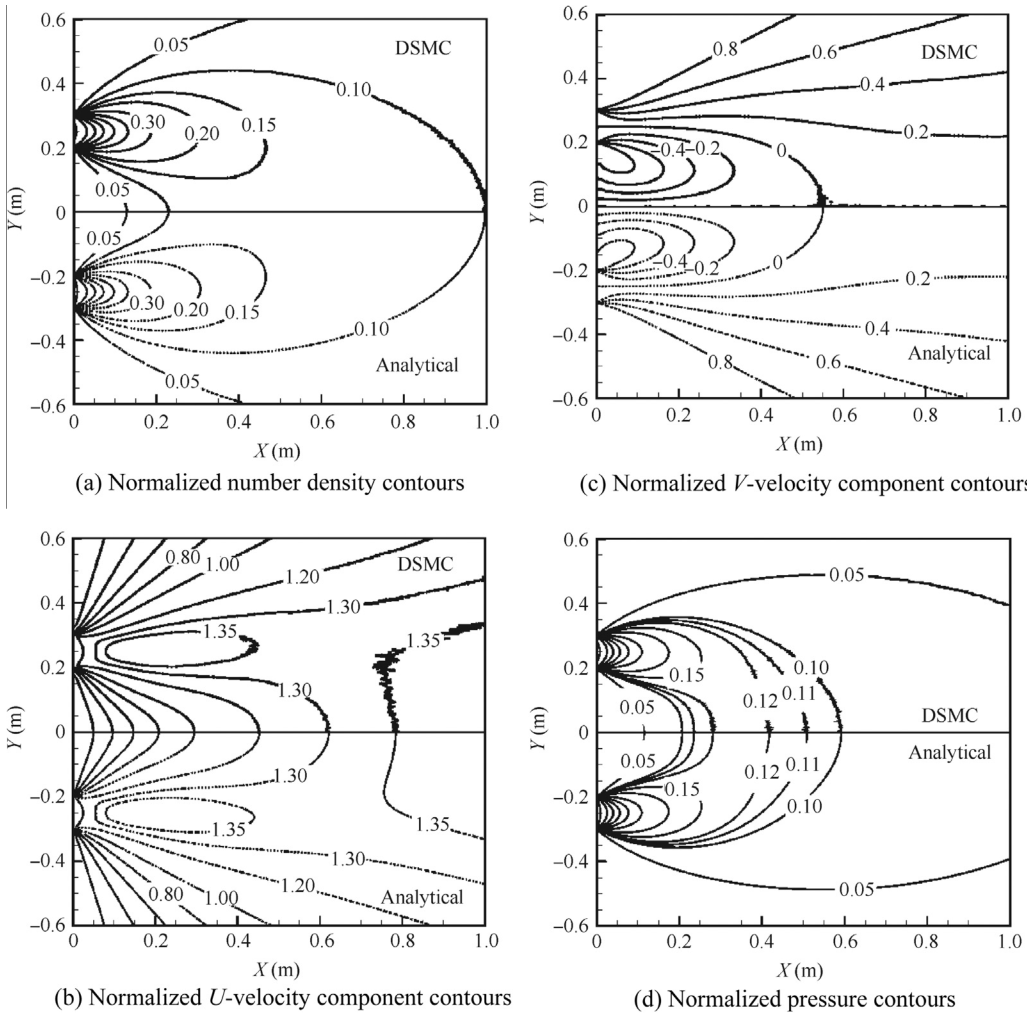

Fig. 5 Comparison of two planar jets, $S_0 = 1.0$.

Fig. 6 shows the results of normalized number density, velocity components and pressure contours inside *XOY* plane, for a cluster of two annular thrusters. From these figures, it is evident that the analytical and DSMC results agree well.

Figs. 7 and 8 are the number density and *U*-velocity component profiles along the two jet centerlines. It shall be emphasized that the Hall thruster simulations are three-dimensional, expensive, and challenging to capture accurate flowfield due to the small annular gap, especially for the pressure results. By comparison, the analytical solutions are economic, accurate and independent of mesh size. These are the merits of the new approach.

Before the final conclusions, some discussions are provided as follows:

(1) For two rarefied planar or annular jets, it is feasible and convenient to compute the combined flowfield properties with a set of relations. The procedure can be conveniently extended to more general or practical situations, e.g., jets or plumes from an array or a matrix of gaseous spray exits.

(2) For the density and velocity components, the computation process is straightforward; however, for the combined pressure results, we shall consider several extra positive terms related to the velocity differences. As such, the traditional treatment using discrete point sources to compute the combined pressure flowfield is questionable.

(3) This approach only utilizes several gaskinetic definitions and previous gaskinetic solutions without involving any numerical schemes, neither particle simulations nor other CFD schemes. Any valid numerical simulation methods shall be able to correctly validate these relations.

5. Conclusions

This paper extends the previous gaskinetic solutions of single rarefied jet to the more realistic situations of multiple rarefied jets into vacuum. The results can also be used to compute for a jet from an exit of arbitrary shapes. The results are of special interest to propulsion. Several DSMC simulations are

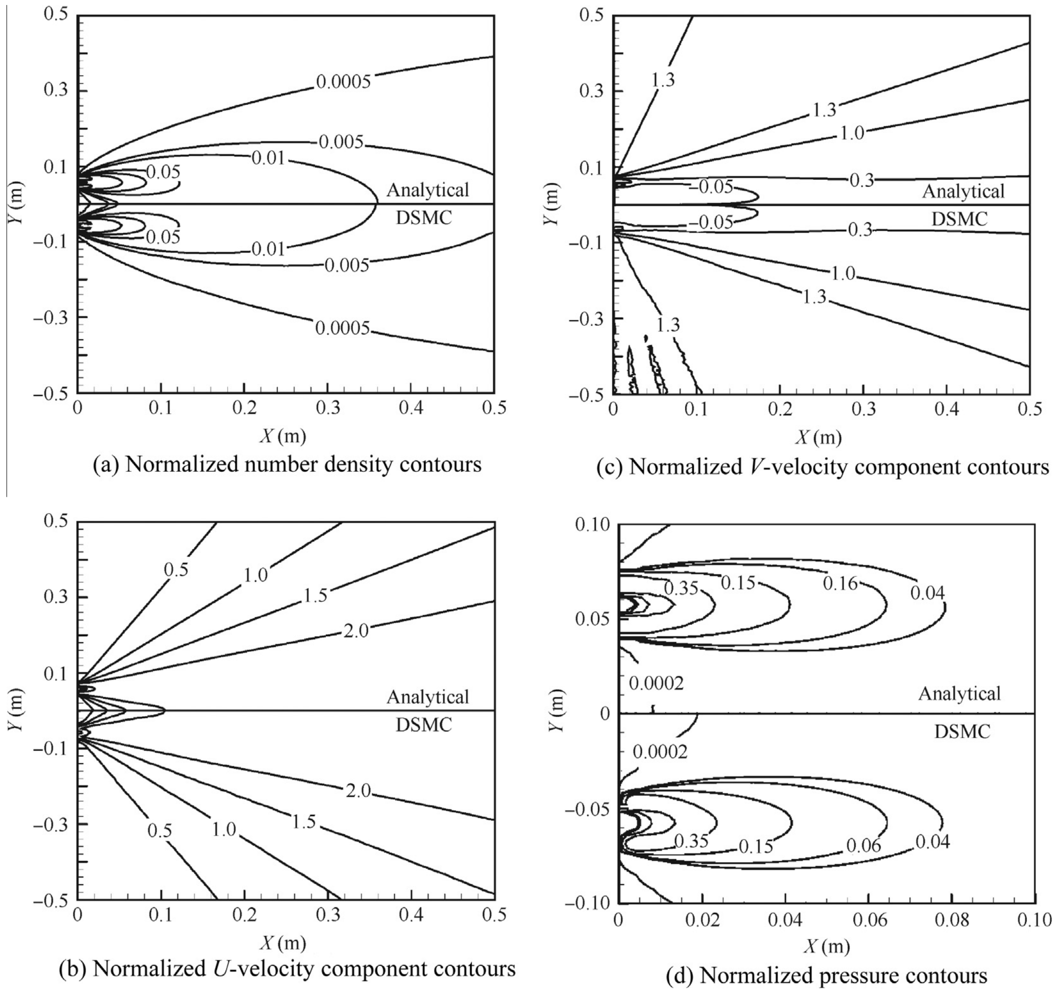

Fig. 6 Normalized number density, velocity components and pressure contours inside *XOY* plane, $Kn = 100$, $S_0 = 2.0$.

performed to validate the analytical formulas, where the characteristic length is the thruster exit width. At high *Kn* number flows, the analytical and numerical simulation results agree well. The flowfield properties of density and velocity components are straightforward to compute, while extra attention shall be practiced to the combined pressure and temperature results. This approach and results are general and are applicable in other high-speed rarefied gaseous flows.

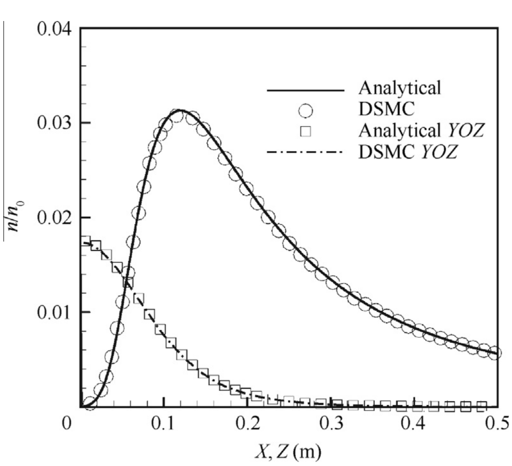

Fig. 7 Two annular jets: normalized number density along the *X*- and *Z*-axises, $Kn = 100$, $S_0 = 2.0$.

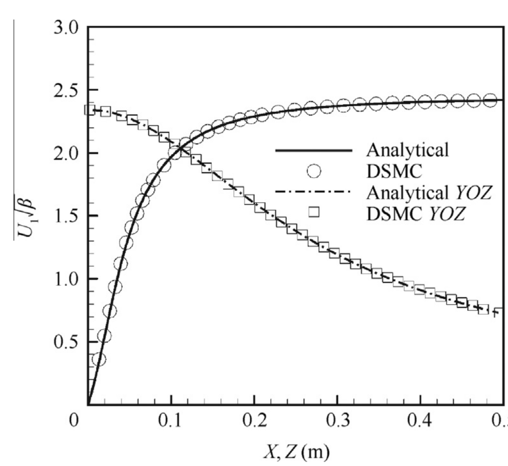

Fig. 8 Two annular jets: normalized *U*-velocity along the *X*- and *Z*-axises, $Kn = 100$, $S_0 = 2.0$.

Acknowledgements

This project is supported by NSF-DMS-0914706, the Research Enhancement Awards from the New Mexico Space Grants, and NSF-CBET-0854411.

References

1. Sanna G, Tomassetti G. *Introduction to molecular beams gas dynamics*. London: Imperial College Press; 2005.
2. Campargue R. Historical account and branching to rarefied gas dynamics of atomic and molecular beams: a continuing and fascinating odyssey commemorated by Nobel Prizes awarded to 23 laureates in physics and chemistry. In: Rarefied gas dynamics: 24th international symposium; 2004. p. 32–46.
3. Haag T. Thrust stand for high-power electric propulsion devices. *Rev Sci Instrum* 1991;**62**(5):1186–91.
4. Rebrov AL. Free jets in vacuum technologies. *J Vac Sci Technol A* 2001;**19**(4):1679–87.
5. Maev R, Leshchynsky V. *Introduction to low pressure gas dynamics spray*. Germany Weinheim: Wiley-VCH; 2008.
6. Bird GA. *Molecular gas dynamics and the direct simulation of gas flows*. New York: Oxford University Press; 1994.
7. LeBeau GJ. A parallel implementation of direct simulation Monte Carlo method. *Comput Methods Appl Mech Eng* 1999;**174**(3–4): 319–37.
8. Simmons FS. *Rocket exhausts plume phenomenology*. Virginia: AIAA Reston; 2000.
9. Simons GA. Effects of nozzle boundary layers on rocket exhaust plumes. *AIAA J* 1972;**10**(11):1534–5.
10. Narasimha R. Collisionless expansion of gases into vacuum. *J Fluid Mech* 1962;**12**(3):294–308.
11. Noller HG. Approximate calculation of expansion of gas from nozzles into high vacuum. *J Vac Sci Technol* 1966;**3**(4):202–7.
12. Brook J. Far field approximation for a nozzle exhausting into a vacuum. *J Spacecraft Rockets* 1969;**6**(5):626–8.
13. Woronowicz M, Ghaffarian S. *Highlights of transient plume impingement model validation and applications*. AIAA-2011-3772; 2011.
14. Cai C, Boyd ID. Theoretical and numerical studies of several free molecular flow problems. *J Spacecraft Rockets* 2007;**44**(3):619–24.
15. Cai C, Boyd ID. Collisionless gas flow expanding into vacuum. *J Spacecraft Rockets* 2007;**44**(6):1326–30.
16. Khasawneh K, Liu H, Cai C. Highly rarefied two-dimensional jet impingement on a flat plate. *Phys Fluids* 2010;**22**(5):1–6.
17. Cai C, Huang X. Rarefied circular jet impingement flows. *AIAA J* 2012;**50**(12):2908–18.
18. Cai C, Wang L. Numerical validations for a set of collisionless rocket plume solutions. *J Spacecraft Rockets* 2012;**49**(1):59–68.
19. Boyd ID, Yim JD. Modeling of the near field plume of a Hall thruster. *J Appl Phys* 2004;**95**(9):4575–84.
20. Cai C. Theoretical and numerical studies of plume flows inside vacuum chambers dissertation. Saarbrucken: Aerospace Engineering, University of Michigan; 2005.
21. Liu H, Cai C, Zou C. GRASP: an object-oriented implementation of the DSMC simulation package. *Comput Fluids* 2012.

Wang Limei is a Ph.D. graduated from the New Mexico State University. She received her B.S. degree from Zhejiang University of Technology in 2007. Her area of research includes hypersonic flow gaskinetic CFD scheme development and rarefied gasdynamics.

Cai Chunpei is an associate professor and Ph.D. advisor at the Department of Mechanical and Aerospace Engineering, New Mexico State University, NM, USA. His current research interests are hypersonic flow, gaskinetic CFD scheme development, rarefied gasdynamics and plasmadynamics, particle simulation methods, space propulsion and space weather.