

Wenhai Li
Department of Mechanical Engineering
Stony Brook University
The PhD Dissertation Topic:

Direct Simulation Monte Carlo Method Technique with Application to Multiple Jets Interaction

Advised by Prof. Foluso Ladeinde

1. Background

With the rapid development of space technology, a lot of emphasis has been placed on the study of the interaction between rarefied free jets. Various jet interaction phenomena exist in the design of spacecraft, an example being multi-nozzle rockets (Fig. 1). The rockets are always equipped with two or more nozzles to provide large impulse and stability. Because of the high altitude, the pressure is low, which cause the plumes from each nozzle to have a large radial extent. Therefore, an interaction between the neighboring plumes may occur. Another example is the spacecraft's Orbiter Reaction Control System (RCS), which always comprises of many primary and vernier engines. The RCS can provide the thrust for altitude maneuvers and for small velocity changes along the orbiter axis by firing the selected engines. If adjacent engines are fired simultaneously, an interaction between the two jets can occur. The jet interaction phenomena also can be seen in the satellite's Altitude Control System (ACS), which is used to control the altitude of the satellite. This system is generally formed by an array of small thrusters. Because the size of the satellite is small and the plume size is large in high altitude, jet interaction between the adjacent plumes can be observed.

The interaction between the jets can greatly change the flow profile, the mixing pattern of the products of combustions and the dynamic of jet impingement. These phenomena can cause a lot of difficulties in the design of a spacecraft. Therefore, jet interaction is important and needs to be investigated in detail. When the interactions take place in the continuum regime, the problem can be solved using the classical continuum method, such as the solution of the Navier-Stokes equations. But when the interactions take place at a high altitude, in which case rarefaction effects become significant, the continuum method will not be applicable. The problem needs to be attacked from the stand point of rarefied gas dynamics (RGD). The Direct Simulation Monte Carlo Method (DSMC) method has been recognized as an extremely powerful technique capable of predicting the rarefied gas flow in the regimes where neither the Navier-Stokes nor the free-molecular approaches are appropriate.

2. Objectives

In this project, the single under-expanded free orifice jet will be simulated by DSMC. The simulation conditions cover from the transition to near-continuum regimes. Both the jet exhausting into vacuum and a finite pressure background will be simulated. The barrel shock and Mach disk structures will be reproduced for the case of near-continuum regimes. The numerical results should be compared with the available experimental results.

After the single jet simulation, the dual interacting jets will be simulated. The formation of the interacting shock wave should be reproduced. The simulation will be

applied for both vacuum and finite pressure background. For the case of finite pressure background, the formation of the Mach disk in the interacting region should be reproduced; therefore the position and diameter of the Mach disk in the interacting region can be studied and compared with the position and diameter of the primary jet. The simulation will be applied for various conditions to analyze the influence of the distance of the orifice separation, the stagnation to background pressure ratio, and the orifice Knudsen number. After the dual jet interaction simulation, the three and four jet interaction problem will be simulated. All of the results will be compared with the available experimental results.

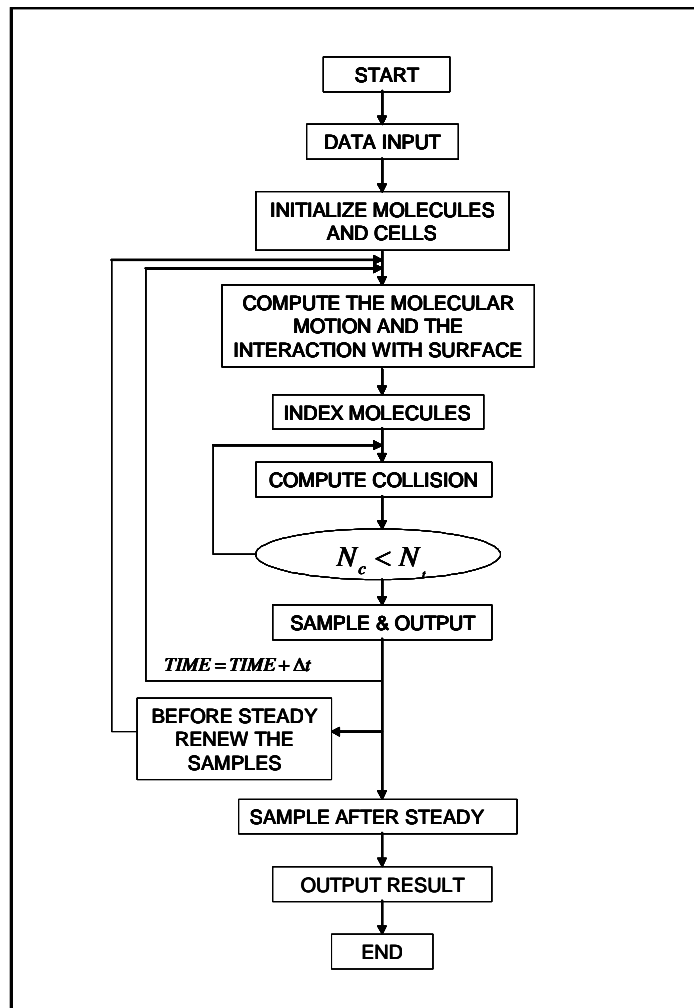
3. Computational Methods and the Code

In this paper, a three-dimensional DSMC code is developed for the simulation of the rarefied gas flows. In this study, the DSMC algorithm is built around the same physical concepts as described by Bird. DSMC is a particle simulation method. It does not solve the Boltzmann equation directly. It solves the Boltzmann equation by mimicking the physical nature of the gas molecular motion and the intermolecular collisions. DSMC is a kind of Monte Carlo method, it uses simulated molecules to substitute for real molecules. A simulated molecule can be regarded as a group of real molecules. DSMC is also a statistical method. It is the collision statistical probability that determines whether or not a pair of molecules will collide, not the distance between the two molecules. This is the biggest difference from the Molecular Dynamics (MD) method, which, on the other hand, is a deterministic method.

The DSMC procedure can be summarized in the following steps:

- (1) Initialization of the program: cells and molecules
- (2) Computation of the flow field: In every time step Δt , the molecular movement and the intermolecular collision are decoupled.
 - Computation of the movement of each molecule with constant velocity.
 - Selection of collision pairs and computation of the intermolecular collisions.
- (3) Sampling the flow field and outputting the result

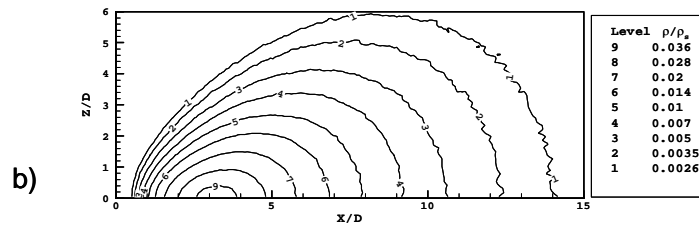
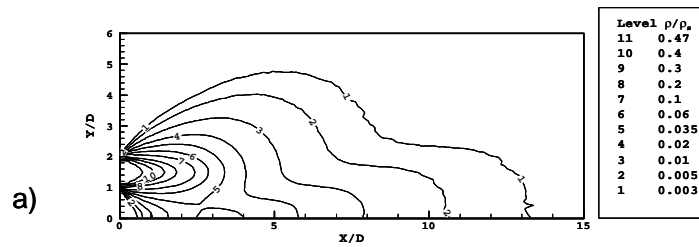
The flow chart for the DSMC code is shown as follows:



The molecular collisions are modeled using the Variable Soft Sphere (VSS) molecular model and the No-Time-Counter (NTC) method is used for the calculation of intermolecular collisions. The discrete internal energy with the temperature dependent collisional number is used in Larsen-Borgnakke (LB) scheme for the inelastic molecular collision.

The code is three-dimensional parallel code which is developed by the author. The code uses the fixed structured mesh and can only work for the fixed domain decomposition. To improve the calculation efficiency, the code will be developed to have the features of the adaptive structured mesh and dynamic domain decomposition.

4. Some Results



This is the density contour for the interacting of the dual jets with the condition: $Kn=0.003$ and $L/D=3$. Where Kn is the orifice Knudsen number, L is the distance of the separation of the two orifices and D is the diameter. For more available results, please check my paper of AIAA-2006-1192.