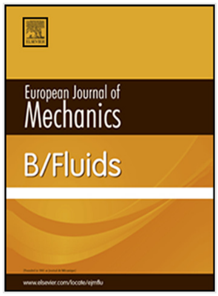

Numerical modeling of high-speed rarefied gas flows over blunt bodies using model kinetic equations

V.A. Titarev

PII: S0997-7546(16)30182-0
DOI: http://dx.doi.org/10.1016/j.euromechflu.2016.12.005
Reference: EJMFLU 3108

To appear in: *European Journal of Mechanics B/Fluids*

Numerical modeling of high-speed rarefied gas flows over blunt bodies using model kinetic equations

V.A. Titarev

Federal Research Center "Computer Science and Control" of RAS, Moscow, Russia

Moscow Institute of Physics and Technology, Moscow, Russia

e-mail: titarev@ccas.ru, titarev.va@mipt.ru

Abstract

The paper is devoted to the development of the numerical approaches to solve model kinetic equations as applied to computing high-speed rarefied gas flows over three-dimensional geometries. The use of a very non-uniform unstructured velocity mesh is proposed, and the influence of the velocity mesh resolution is studied in detail. The test problems include a simple flow over a planar circular cylinder and a three-dimensional flow over a model winged re-entry vehicle.

Introduction

At present, the Direct Simulation Monte-Carlo method is most popular approach to modeling of high-speed rarefied gas flows [1,2]. An alternative is provided by the direct numerical solution of the Boltzmann kinetic equation for velocity distribution function with either exact or model (approximate) collision integral. The deterministic nature of the equation allows creating efficient implicit high-order methods for both stationary and time-dependent flows. In recent years, there has been significant progress in developing numerical methods to solve kinetic equations in three space dimensions, see e.g. [3–5]. However, most applications have been in the area of the microchannel flows, whereas modeling of the high-speed rarefied flows remains challenging.

The main obstacle in using kinetic equations is the number of the molecular velocity nodes grows rapidly with the free-stream Mach number if a conventional structured velocity mesh is used. This results in very high requirements in computer memory and prohibitively large computing times. The authors of [6, 7] proposed to use an octree-type velocity mesh, which is automatically refined where needed. In particular, the latter work is devoted to the external high-speed flows and includes examples of the calculations. However, the approaches based on octree-type adaptive meshes are in general more complicated as compared to conventional meshes and require additional steps in the solution procedure. There is also no strict comparison of the results