

Effects of Continuum Breakdown on the Surface Properties of a Hypersonic Sphere

Timothy D. Holman* and Iain D. Boyd†
 University of Michigan, Ann Arbor, Michigan 48109

DOI: 10.2514/1.43509

This study investigates the effects of continuum breakdown on the surface aerothermodynamic properties (pressure, shear, heat transfer rate) of a sphere in Mach 10, 25, and 45 flows of nitrogen gas in regimes varying from continuum to rarefied gas. A rotational energy relaxation model is employed in the computational fluid dynamics code and is tested to confirm its accuracy. As the global Knudsen number is increased, from continuum flow to a rarefied gas, the amount of continuum breakdown seen in the flow and on the surface is increased. This increase in continuum breakdown affects the surface properties, such that an increase in the differences between computational fluid dynamics and direct simulation Monte Carlo method is observed. As the Mach number is increased, the amount of continuum breakdown observed in the flow is increased, but the gradient length local Knudsen number stays approximately constant. Even though the amount of continuum breakdown has increased, the difference between computational fluid dynamics and direct simulation Monte Carlo method remains relatively constant. The last part of this study compares the results of the sphere with that of the analogous cylinder case. At the same global Knudsen number, the differences in the surface properties between computational fluid dynamics and direct simulation Monte Carlo method increase when the simulation is run axisymmetrically.

Nomenclature

C_p	=	pressure coefficient
C_q	=	heat flux coefficient
C_τ	=	shear coefficient
\bar{c}	=	mean speed, m/s
d	=	molecular diameter, m
E	=	energy per unit volume, J/m ³
Kn_{GLL}	=	gradient length local Knudsen number
Kn_∞	=	global Knudsen number
k	=	Boltzmann constant, 1.38×10^{-23} J/K
L	=	characteristic length, m
M	=	Mach number
m	=	mass, kg
m^*	=	relative mass, kg
P	=	average probability
p	=	pressure, Pa
q	=	heat transfer rate, W/m ²
Re	=	Reynolds number
T	=	translational temperature, K
U	=	freestream velocity, m/s
Z	=	collision number
γ	=	ratio of specific heats
ζ	=	internal degrees of freedom
λ	=	mean free path, m
μ	=	viscosity, Pa · s
ν	=	mean collision rate, 1/s
ρ	=	density, kg/m ³
σ	=	collision cross section, 5.81×10^{-21} m ² , accommodation coefficient
τ	=	shear stress, Pa, relaxation time, s
ω	=	variable hard sphere temperature exponent

Subscripts

cont	=	continuum
LT	=	Landau–Teller
p	=	park
part	=	particle
r	=	rotation
ref	=	reference value
v	=	vibration
∞	=	freestream

I. Introduction

WITH a renewed desire to send humans back to the moon and beyond there is a need for accurate studies of the flow behavior over hypersonic vehicles to precisely determine how they will perform when entering an atmosphere. Experiments and flight tests are extremely difficult and expensive, and so there is need for computational models that can be used for design and development of hypersonic vehicles.

A hypersonic vehicle that is entering an atmosphere will go through many different flow regimes due to the change in atmospheric density with altitude. The flow can be characterized by the following Reynolds and the Knudsen numbers:

$$Re = (\rho UL)/\mu \quad Kn = (\lambda/L) \propto (1/\rho L) \quad (1)$$

The flow can be considered continuum when the Knudsen number is much less than one. In the continuum regime, flows should be simulated using traditional computational fluid dynamics techniques by numerically solving the Navier–Stokes equations. However, when the Knudsen number becomes larger, the continuum assumption in the Navier–Stokes equations starts to breakdown. This is due to the fact that these equations are derived from kinetic theory based on the assumption of small perturbations from an equilibrium velocity distribution function [1]; therefore, computational fluid dynamics (CFD) only works in near equilibrium flows. When the Knudsen number grows, only a noncontinuum technique can be used, such as the direct simulation Monte Carlo (DSMC) method [2]. DSMC is a Monte Carlo particle method for simulating nonequilibrium gas flows. DSMC is required for accurate flow analysis of hypersonic rarefied flows for which the continuum flow equations are invalid and can be used in any dilute gas flow. Unfortunately, DSMC is about an order of magnitude more expensive than traditional CFD methods

Presented as Paper 3928 at the 40th AIAA Thermophysics Conference, Seattle, WA, 23–26 June 2008; received 29 Jan. 2009; revision received 2 June 2009; accepted for publication 14 June 2009. Copyright © 2009 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved. Copies of this paper may be made for personal or internal use, on condition that the copier pay the \$10.00 per-copy fee to the Copyright Clearance Center, Inc., 222 Rosewood Drive, Danvers, MA 01923; include the code 0887-8722/09 and \$10.00 in correspondence with the CCC.

*Graduate Student, Department of Aerospace Engineering, Member AIAA.

†Professor, Department of Aerospace Engineering, Associate Fellow AIAA.