International Journal of Heat and Mass Transfer 57 (2013) 657–661

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Heat release effects on drag reduction in high speed flows

A.M. Tahsini

Aerospace Research Institute, Tehran, Iran

ARTICLE INFO

Article history:
Received 13 June 2012
Received in revised form 14 October 2012
Accepted 22 October 2012
Available online 23 November 2012

Keywords:
Chromium
Numerical simulation
Reaction
Drag reduction
Hypersonic flow

ABSTRACT

Heat addition due to exothermic reactions of ablated chromium in hypersonic flow on coated blunt bodies is numerically investigated. The advection upstream splitting method is used as computational scheme. Conjugate heat transfer is considered by simultaneous solution of flow and solid phase governing equations to compute the surface regression rate. The results show that using such coatings on blunt noses in high speed flows will increase the shock standoff distance and decrease the aerodynamic drag but not so much that previously reported. Using these simulations, the chromium proper thickness will be estimated more precisely for specific flight mission.

1. Introduction

Using blunt nose configurations to reduce the aerodynamic heat loads on hypersonic vehicles causes significant enhancement in wave drag which occupies about two thirds of the total drag. Significant reductions of hypersonic wave drag should result in considerably smaller propulsion system requirements, reduced fuel consumption, and considerably larger payloads at smaller take-off gross weight. There have been different methods suggested to decrease such drag forces encountered by the high speed projectiles. These techniques are using spikes, energy deposition and counterflow supersonic jets which have been analyzed experimentally or numerically in past decade. The spike ahead of the blunt body acts like a flow separator and creates a recirculating region ahead of the blunt cone, where the pressure will be lower than the stagnation zone. Therefore the spike dramatically reduces the wave drag by creating a low dynamic-pressure flow separation [1–3]. The heat deposition is achieved by many different means, such as hot gas, laser beams, ultraviolet light, or electric arc and plasma beams, and creates a low-density core through which the vehicle can travel, thereby significantly reducing the drag force [3–5]. The supersonic jet emanating from the blunt body interacts with the oncoming flow causing the bow shock wave to stand away from the surface and takes the form of a new body that consists of the original body with a spike with a boundary defined by the interface between the jet and the mainstream. So the wave drag reduction is derived from both the splitting of a single strong shock into multiple shock waves and replacing the blunt body by a slender equivalent body [6,7]. Although these techniques have shown an appreciable reduction in the wave drag, their application to practical systems is inhibited by the enhancement in system complexity or other problems. As an example, although the spike can reduce the drag significantly, it needs to endure the large heat flux at the apex, and the pitching moment generated in flights with angle of attack.

The most recent developed drag reduction technique is heat addition through the ablated surface, which is analyzed experimentally [8]. It is practically simple method where the heat is added through the exothermic reaction of the ablated atoms from the nose portion coated with thin film of appropriate material. Based on current information, computational study for this novel idea has not been performed before. In the present study, the numerical simulation of such problems has been done to investigate the heat release drag reduction phenomena using the ablative surfaces on blunt cones in hypersonic flows.

2. Numerical simulation of governing equations and validation

The conservation equations of mass, momentum, energy and chemical species to study the compressible reactive flow in an axisymmetric geometry are used. These equations in conservation form are presented here [9].

$$\frac{\partial U}{\partial t} + \frac{\partial (F + F_v)}{\partial x} + \frac{\partial (G + G_v)}{\partial y} + \frac{G + G' + G_v + G'_v}{y} = ST \quad (1)$$

E-mail address: a_m_tahsini@yahoo.com

0017-9310/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.10.064

Nomenclature

c_v	constant volume specific heat	u	axial velocity
D	mass diffusion coefficient	v	radial velocity
e	total specific energy	$\Delta h^o_{f,j}$	enthalpy of formation of species j
h	total enthalpy	η	normal coordinate in solid phase
H_s	solid phase heat of vaporization	μ	dynamic viscosity
k	thermal conductivity	ρ	gas phase local density
m_j	mass fraction of species j	ρ_s	solid phase density
p	local pressure	τ_{ij}	stress tensor
q	heat flux	$\dot{\omega}_j$	mass rate of production of species j
$\dot{r}$	solid phase regression rate		
T	local temperature		

where

$$U=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho e\\ \rho m_j\end{bmatrix}\quad F=\begin{bmatrix}\rho u\\ \rho uu+p\\ \rho uv\\ \rho uh\\ \rho um_j\end{bmatrix}\quad G=\begin{bmatrix}\rho v\\ \rho vu\\ \rho vv+p\\ \rho vh\\ \rho vm_j\end{bmatrix}\quad G'=\begin{bmatrix}0\\0\\-p\\0\\0\end{bmatrix}$$
$$F_v=\begin{bmatrix}0\\ -\tau_{xx}\\ -\tau_{xy}\\ q_x-u\tau_{xx}-v\tau_{xy}\\ -\rho Dm_{j_x}\end{bmatrix}\quad G_v=\begin{bmatrix}0\\ -\tau_{yx}\\ -\tau_{yy}\\ q_y-u\tau_{yx}-v\tau_{yy}\\ -\rho Dm_{j_y}\end{bmatrix}\quad G'_v=\begin{bmatrix}0\\0\\ \tau_{\theta\theta}\\0\\0\end{bmatrix}\quad ST=\begin{bmatrix}0\\0\\0\\0\\ \dot{\omega}_j\end{bmatrix} \tag{2}$$

The stress terms are defined as:

$$\tau_{xx}=\mu\left(2\frac{\partial u}{\partial x}-\frac{2}{3}\nabla.V\right)\quad \tau_{xy}=\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)$$
$$\tau_{yy}=\mu\left(2\frac{\partial v}{\partial y}-\frac{2}{3}\nabla.V\right)\quad \tau_{\theta\theta}=\mu\left(2\frac{v}{y}-\frac{2}{3}\nabla.V\right) \tag{3}$$

Here

$$e=c_vT+(uu+vv+ww)/2+\sum m_j\Delta h^o_{f,j}$$
$$q=-k\nabla T-\rho D\sum h_j\nabla m_j \tag{4}$$
$$h_j=\Delta h^o_{f,j}+\int c_{p_j}dT$$

The transport coefficients for the species are obtained by application of kinetic theory, and mixing rule is used for the mixture. The data of specific heats and formation enthalpies for each species are given by JANAF thermo-chemical tables.

Heat transfer into the solid phase is simulated using an unsteady one dimensional energy equation coupled with gas phase equations; conjugate heat transfer problem is considered as shown in Fig. 1. The regression rate is calculated from the energy balance in gas-solid interface.

$$\frac{\partial T}{\partial t}=\alpha_s\frac{\partial^2 T}{\partial \eta^2}+\dot{r}\frac{\partial T}{\partial \eta}$$
$$q_g=q_s+\rho_s\dot{r}H_s \tag{5}$$

Here, the cell centered finite volume method is used to discrete the gas phase equations. The time integration is accomplished by an explicit time stepping scheme [10]. Viscous terms are calculated using a central scheme and inviscid terms are treated using an AUSM+ method to express the numerical flux on each cell faces [11]. The developed numerical program has been validated using variety of benchmark problems and experimental data, and used satisfactorily to study the different reactive flows [12–14]. Some related examples are explained here. It should be noticed that the grid independency studies have been done in all numerical simulations presented.

2.1. Supersonic flow on sphere and cylinder

The numerical scheme has been tested for the calculation of the shock standoff distances of a cylinder and a sphere in supersonic flow. Shock standoff distance is a non-dimensional parameter

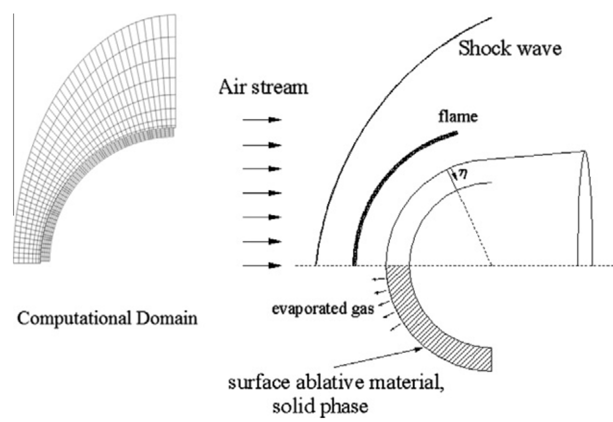

Fig. 1. Schematic configuration of problem and computational domain.

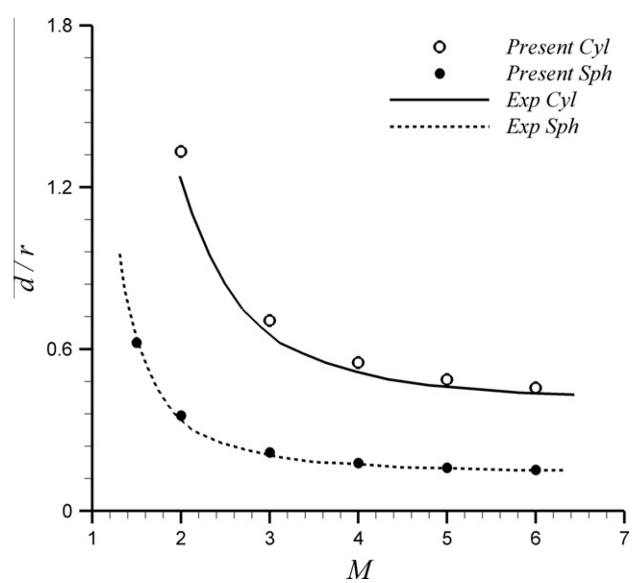

Fig. 2. Shock standoff distances for sphere and cylinder in supersonic flow.