

# SIMULATIONS OF VISCOUS HYPERSONIC DOUBLE-CONE FLOWS:

## INFLUENCE OF NUMERICS

Marie-Claude Druguet\*

*Polytech'Marseille - Mécanique Energétique  
UMR 6595 CNRS / Université de Provence  
13453 Marseille Cedex 13 - FRANCE*

Graham V. Candler<sup>†</sup>

Ioannis Nompelis<sup>‡</sup>

*Aerospace Engineering and Mechanics & Army HPC Research Center  
University of Minnesota  
Minneapolis MN 55455 - USA*

### Abstract

This paper presents the results of a study on the effects of the numerics on the simulation of a hypersonic flow past a sharp double-cone. Previous studies have shown that the double-cone flow is challenging to compute, making it interesting for testing both numerical schemes and physical models. In the present study we focus on the numerical aspects only, and show that the results are very sensitive to the numerical flux evaluation method used. However, when the grid is fine enough, all the flux evaluation schemes give the same results, though this may require a very large number of points for the most dissipative schemes. On the other hand, the most accurate schemes have the great advantage of giving the final results on a coarser grid, and are therefore cheaper. Interestingly, this study also shows that the modified Steger-Warming solver gives as accurate results as other well-known accurate schemes, such as the Roe scheme. Finally, it is important to point out that this work could not have been done without an implicit code.

### Introduction

The flow generated by a double-cone geometry is an interesting CFD test case to study, because it can reproduce complex phenomena taking place in flows past future hypersonic vehicles. These vehicles will likely have regions of separated flow, for example near control surfaces and behind flame holders. Additionally, shock-shock interactions and shocks impinging

on the vehicle surface cause high localized aerothermal loads. Recent work<sup>1-4</sup> has shown that these flows are challenging to compute. The attached leading edge shock wave interacts with a detached bow shock wave formed from the second cone, and this interaction produces a transmitted shock wave that impacts the second cone wall. This impact produces very high surface pressures and heat transfer rates on the second cone. Because of the high pressures at the impact location, the flow separates near the cone-cone juncture and a recirculation zone develops, which in its turn alters the shock interaction. The size of the separation zone is very sensitive to the shock angles and to the strength of the shock interaction. Downstream of the shock impingement location, a supersonic jet develops along the wall of the second cone, within the subsonic shock layer. Figures 1 to 3 – temperature, Mach number and pressure contours – show the main characteristics of this flow configuration.

The present work is related to the NATO Research and Technology Organization (RTO) Working Group 10 effort on CFD code validation. The double cone flow is one of their test cases and was designed to ensure a laminar viscous-inviscid interaction. The free-stream flow conditions have been designed so that the flow may be simulated by a non-continuum approach (direct simulation Monte Carlo - DSMC) as well as a continuum one (Navier-Stokes). Comparisons made to date between numerical results and experimental data have shown that both methods are capable of predicting most of the flow, even though these near-continuum flows are extremely difficult for the DSMC method, and provided that there is sufficient grid resolution for the NS approach. Both methods gave comparable computed results. Comparisons of computed

---

\* Chargée de Recherche CNRS, Member AIAA  
(druguet@polytech.univ-mrs.fr)

† Professor, Associate Fellow AIAA (candler@aem.umn.edu)

‡ Graduate Research Assistant, Student Member AIAA

Copyright © 2003 by Marie-Claude Druguet. Published by the American Institute of Aeronautics and Astronautics, Inc. with permission.