6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11-15 June 2018, Glasgow, UK

Unsteady Simulations of Nitrogen Flow over a Double Cone

Rowan J. Gollan[1] and Peter A. Jacobs[1]

[1] Centre for Hypersonics, School of Mechanical & Mining Engineering, The University of Queensland, 4072, Australia, email: r.gollan@uq.edu.au

Keywords: *hypersonics, nonequilibrium flows, high-enthalpy flows, separated flows, shock wave boundary layer interactions, CFD*

In estimating the heating on hypersonic vehicles, shock wave boundary layer interactions are of interest because there is a local augmentation of the surface heating rate in the region of boundary layer reattachment. Present-day computational fluid dynamics tools (CFD) have difficulties predicting the separation zone and associated heating, with modelling difficulties that include nonequilibrium thermochemistry and unsteady flow dynamics. [1]. As such, there is interest in advancing the predictive capabilities of CFD so that reliable estimates of heating may be used for vehicle design. A set of experiments presented by Nompelis et al. [2] used a double-cone geometry with nitrogen test gas and were designed to investigate the effect of varying the total enthalpy of the free stream such that high-temperature gas effects are expected to have some influence. These experiments provide a useful validation case for nonequilibrium modelling in hypersonic flows.

A complication with these double cone experiments is that experiments and simulations disagree about whether the flow field has a steady state or is inherently unsteady for certain conditions [1]. In this paper, we present the results of simulating the flow of nitrogen over a double cone in an unsteady, time-accurate manner. The simulations have been performed using the hypersonic flow solver, Eilmer [3]. We show that the unsteadiness is only apparent on very fine grid resolutions. When a time-average is taken of the unsteady results, the best match to experiment is found in terms of the separation location and the heat loads.

REFERENCES

[1] Knight D, Longo J, Drikakis D, Gaitonde D, Lani A, Nompelis I, Reimann B, and Walpot L. Assessment of CFD capability for prediction of hypersonic shock interactions. *Progress in Aerospace Sciences* 2012; **48**:8–26.

[2] Nompelis I, and Candler G. Numerical investigation of double-cone flow experiments with high-enthalpy effects. *AIAA Paper 2010-1283*, American Institute of Aeronautics and Astronautics 2010.

[3] Gollan R, and Jacobs P. About the formulation, verification and validation of the hypersonic flow solver Eilmer. *International Journal for Numerical Methods in Fluids* 2013; **73**(1):19–57.