Aerospace Science and Technology 58 (2016) 48–59

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Rapid prediction of hypersonic blunt body flows for parametric design studies

W. Schuyler Hinman *, Craig T. Johansen

ARTICLE INFO

Article history:
Received 9 May 2016
Received in revised form 3 August 2016
Accepted 4 August 2016
Available online 10 August 2016

ABSTRACT

The present work discusses the practical advantages and disadvantages of using simplified numerical methods and computational fluid dynamics in parametric design studies of hypersonic blunt bodies. Similarly, the advantages of using problem-specific simplifications to the governing equations to reduce computational cost are discussed. The uncertainty associated with using various methods to analyze hypersonic blunt body flows has been quantified through comparison to numerical solutions of the compressible Navier–Stokes equations. In particular, selected methods that are well defined in the literature, such as the modified Newtonian method, transformed finite difference grids, and the method of characteristics in the supersonic region, have been utilized to solve two cases of interest. An improvement to the prediction methods has been achieved through the inclusion of an iterative interaction between the boundary layer displacement thickness and the external inviscid free-stream. Results were collected for accuracy and computing time for each method including under-resolved compressible Navier–Stokes simulations. The collective information was used as a case-study to discuss the balance an engineer must find between simulation fidelity, resolution, accuracy, simulation time, and development time.

1. Introduction

The design of hypersonic blunt-body vehicles requires accurate predictions of aerodynamic loads such as drag, friction, moments, and heat flux. In general, computational fluid dynamics (CFD) simulations of the Navier–Stokes equations provide the most accurate and detailed predictions of these flow parameters. The highly accurate results of Navier–Stokes simulations are particularly useful when a vehicle geometry is past the initial design phase and more accurate predictions are needed to optimize re-entry trajectory, heating or placement of control mechanisms. However, in the ongoing advancement and development of blunted hypersonic geometries, the shape of the body may be further improved upon through parametric studies and optimizations [1–3]. These types of studies involve evaluating many design variables. In these cases, where the best results are achieved with large populations, CFD is an inefficient prediction tool due to the high computational cost. Often a compromise is made in grid resolution, and hence accuracy, in order to keep a parametric or optimization study feasible [4]. It is unclear whether such a reduction in resolution in order to reduce computing time, while maintaining the highest fidelity governing equations, is the appropriate strategy. Alternatively, the governing equations themselves can be simplified and solved with high resolution in order to achieve a reduction in computing time, provided that some prior wisdom of the problem exists. Both of these approaches result in decreased accuracy either by a lack of resolution, or from simplifications to the governing equations. A major problem in starting an analysis or optimization campaign is "how does the designer/engineer/scientist choose the correct tool?". In flow prediction, the engineer must weigh all of the costs of available tools against their value in terms of speed, reliability and accuracy. Obvious costs include the acquisition/development costs and the operational costs. However, the learning curve associated with a new tool is also an important cost. These decisions are often made based on previous experiences and expertise. The purpose of this work is to assist with making that decision by systematically assessing the accuracy and cost of a range of analysis methods used in hypersonic flow prediction. Assessment is based on the prediction of a complex, but well-known, canonical flow problem. Hypersonic flow over a cylinder involves regions that are described by elliptic, parabolic, and hyperbolic equations. Large changes in temperature require proper treatment of the thermophysical properties. The presence of shock waves introduce discontinuities that compromise accuracy, lead to instability, and increase computing time for most standard numerical methods. The present work has attempted to examine this problem by comparing a series of blunt body flow calculations of varying fidelity. The study includes two test cases: (1) a Mach 6, two-dimensional, laminar flow over a circular cylinder with an

* Corresponding author.
E-mail address: wshinman@ucalgary.ca (W.S. Hinman).

http://dx.doi.org/10.1016/j.ast.2016.08.007
1270-9638/

Nomenclature

a	Velocity profile parameter (in Equation (6) and (7)), also speed of sound (m/s)
b	Body shape, measured radially from the origin (m)
As	Sutherland constant
C	$C = \frac{\rho\mu}{\rho_e \mu_e}$
C_p	Coefficient of pressure
C_1, C_2, C_3	Coefficients for Kays' laminar heating (in Equation (3))
c_p	Specific heat capacity J/(kg K)
c_f	Coefficient of friction
F	$F = \frac{U}{U_e}$
G	$G = \frac{H}{H_e}$
G_e	$\rho_e U_e$ (in Equation (3))
H	Total enthalpy (J/kg)
H, J, R, P	Boundary layer integral parameters (in Equations (6) and (7))
m	Displacement thickness growth ($\frac{d\delta^\star}{dx}$)
M	Mach number
p	Pressure (Pa)
q_∞	Free-stream dynamic pressure (Pa)
Pr	Prandtl number
r	Adiabatic recovery factor (in Equation (20)), also relaxation factor (in Equation (26))
R	Local axi-symmetric radius of curvature (m) (in Equation (3))
R_g	Specific gas constant J/(kg K)
Re	Reynolds number
Re_c	Cell Reynolds number
$\bar{s}$	Levy–Lees stream-wise coordinate
T	Temperature (K)
T_{aw}	Adiabatic wall temperature (K)
T_s	Sutherland's temperature (K)
u	Velocity vector (m/s)
U	Tangent velocity component (m/s) (boundary layer equations)
V	Normal velocity component (m/s) (boundary layer equations)
X	Stewartson stream-wise coordinate
Y	Stewartson stream-normal coordinate
γ	Specific heat ratio
δ^*	Boundary layer displacement thickness (m)
η	Levy–Lees stream-normal coordinate
θ	Deflection angle (radians or degrees)
μ	Dynamic viscosity (kg/ms)
ρ	Density (kg/m^3)
τ	Shear stress (N/m^2)

Subscript

e	Property at boundary layer edge
i	Transformed quantity, also iteration
o	Property at forward stagnation point
w	Property at wall
∞	Free-stream property

adiabatic wall, not including the separated wake region, and (2) a Mach 10, two-dimensional, laminar flow over a blunted leading edge with a cold wall.

Parametric studies and design optimizations are a fundamental tool in the development of future aerospace technologies. In CFD applications these types of studies can be very expensive due to the high computational requirements of the governing Navier–Stokes equations. Because of this there is interest in finding ways to reduce this cost with minimal compromise in accuracy and confidence. For example, surrogate based analyses are very common in CFD applications because they do not require a Navier–Stokes simulation to be performed at every data point. Instead, a surrogate model uses data achieved through high fidelity simulations performed at select points in the design space to create a model [5]. Additionally, surrogate models can be constructed using a combination of high and low fidelity simulations [6,7]. This leads to a significantly reduced cost of performing parametric optimizations and analyses. However, even in a surrogate-model based study, a complicated design space can still require numerous high-fidelity simulations to be performed. Additionally, it is important to understand the limitations and benefits of using intermediate fidelity approaches. For this reason there is still value in reducing the cost of individual simulations. It is easy to assume that high-fidelity CFD is required in order to produce the level of accuracy necessary in modern design. While in some cases this is true, in others it is worth investigating the possible reduced-order and simplified models that have been developed and used in the past. For example, in previous work by the present authors, simplified numerical solutions were used with a genetic algorithm to optimize waverider leading edges [1]. The reduced-order model allowed a large number of simulations to be performed with high accuracy. The optimized results were then examined using high-fidelity CFD and it was found that the predicted performance gains from the optimization were not significantly in error. Because of the speed and accuracy of the models used, no surrogate based modeling was required.

Many simplified solutions to flow over hypersonic blunt bodies exist. A representative few are presented here in order to provide the necessary data for analysis and discussion. Two methods of varying accuracy for inviscid flow have been examined: (1) The modified Newtonian method [8] and (2) a numerical solution to the Euler equations using finite differences [8,9] combined with the method of characteristics [8,10]. Similarly, several methods for producing boundary-layer flow solutions are examined: (1) A simple solution for convective heat flux described by Kays et al. [11], (2) a solution to the integral boundary layer equations using the Cohen–Reshotko family of profiles [12,13], and (3) a direct numerical solution of the complete compressible boundary layer equations [14]. Each of these methods are solved in combination with each other to achieve a representative variation in accuracy. These methods are described in more detail in Section 2. Because flowfields of practical interest are a mixture of viscous and inviscid flow it is often necessary to account for viscous-inviscid interaction. In the case a circular cylinder an interaction occurs between the shoulder expansion fan and the boundary layer flow. A simple iterative algorithm to account for this interaction is presented in Section 3.

For the solution of the compressible Navier–Stokes equations the open source CFD package OpenFOAM version 2.3.0 was used [15]. The solver used in this work, rhoCentralFoam, has been compared to experiment as well as other solvers from other CFD packages and shown acceptable results [16–19]. Arisman et al. [17,18] used a modified form of the rhoCentralFOAM solver to compute the injection of nitric oxide in a cross-flow configuration into a Mach 10 boundary layer with air as the freestream. Vertical distributions of predicted streamwise velocity were compared to experimental planar laser-induced fluorescence (PLIF) molecular tagging velocimetry (MTV) measurements. In addition to velocity, a surface