Aerodynamic Heat Prediction on a 15 degree Cone-Cylinder-Flare Configuration Using 2D Axisymmetric Viscous Transient CFD

Weerawut Charubhun and Pawat Chusilp
Aeronautical Engineering Division
Defence Technology Institute
Pakkret, Nonthaburi, Thailand
weerawut.c@dti.or.th, pawat.c@dti.or.th

Abstract—**This paper investigates a method to predict aerodynamic heating on a supersonic flare body using a 2D axisymmetric transient viscous CFD simulation. Five turbulent models and three simulation time step sizes were utilized. The surface temperature of the rocket predicted by the simulations was compared to the published experimental data obtained from the first 26 s of a rocket flight test. It was found that the Transition SST model performed better than the others. Furthermore, the simulation results matched the experimental data well during the first 7 second. Then the difference between the simulation results and the experimental data became noticeable. But their trends were still the same and CFD simulation overestimated the temperature most of the time.**

Keywords—Aerodynamic heating; Supersonic flow; Fransient CFD simulation; Skin temperature.

I. Introduction

When a rocket travels at supersonic speed or higher through Earth atmosphere, the air molecules adjacent to the projectile skin are brought to rest and the kinetic energy is consequently converted to heat so called "Aerodynamic Heat." Accurate prediction of the aerodynamic heat is essential when designing a proper thermal protection system to protect heat sensitive components especially those electronic components in the front section, such as telemetry or guidance module, and explosive materials, such as TNT. Many researchers have proposed engineering formulae and calculation methods to predict aerodynamic heat for supersonic and hypersonic vehicles. Fay and Riddell [1] proposed formulae to compute local heat transfer coefficient. Quinn and Palitz [2] presented formulae for heat transfer rate and skin temperature for small-zero pressure gradient surfaces. Eckert [3, 4] introduced "Reference Enthalpy Method" to estimate heat transfer rate. Hazen [5] and Zoby [6] proposed formulae for wall enthalpy and convective heating in hypersonic flow. Accuracy of the results from these methods depend heavily on accurate heat transfer coefficients and heat transfer rate that is changed with flight conditions, skin material and rocket geometry. Without data from flight tests, accurate values of these parameters can be hardly obtained.

Since the emerging of computational science and engineering, Computational Fluid Dynamic (CFD) has been introduced to analyze aerodynamic heating. However, a full viscous CFD simulation on a very long flight time of hypersonic vehicles was rarely practical as it needs high mesh quality and small time step size and hence very long computation time. On the other hand, inviscid CFD simulation, which requires much less computation time but less accurate, was usually utilized in combination with engineering calculation method to enhance the accuracy. Theodore and Xiaolin [7] included 5^{th} order shock capturing scheme in inviscid CFD simulation to improve the prediction accuracy for hypersonic vehicles. Christopher and William [8] introduced inviscid-boundary layer method to compute the surface heating rates at selected points over X-34 aircraft. Duarte and Silva [9] proposed a calculation procedure involving inviscid CFD and engineering calculation for missile or rocket conceptual design phase.

With greater processing power of modern computers, two dimensional viscous transient CFD simulations to analyze aerodynamic heating on the whole flight trajectory of supersonic projectiles has become possible. For example, aerodynamic heating analysis of the X-15 mid-wing span geometry using viscous CFD simulation were presented by Mukkarum [10].

This paper investigates the application of two dimensional (2D) viscous transient CFD simulation to predict aerodynamic heat on a 15° Cone-Cylinder-Flare rocket configuration when it travels up to 4.7 Mach. The simulation results were compared to the experimental data published by NASA Langley Research Center [11]. Different turbulent models and time step sizes were utilized and the accuracy of results of each setting were evaluated. The following sections are structured as follows. Section 2 explains the problem background and describes all simulation cases. Section 3 describes the application of CFD simulation on aerodynamic heating problem in this paper. The simulation results were presented and compared to the experimental data in Section 4. Finally, Section 5 draws a conclusion.

II. Problem Description

Temperature data measured on the surface of the 15° Cone-Cylinder-Flare rocket during the flight test conducted by NASA Langley Research Center [11] were used as a