40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit
11 - 14 July 2004, Fort Lauderdale, Florida

AIAA 2004-3933

Modeling and Simulation of Dynamic Inlet Flow Distortion Generation

Keith P. Savage,* Gary A. Flandro,† and Joseph Majdalani‡
University of Tennessee Space Institute, Tullahoma, TN 37388

Integrating a jet engine into an airframe involves many design and testing steps. Furthermore, the development of high performance fighter aircraft has added new requirements to this process. Ground test facilities are used to simulate the effect of inlet shape on the performance of an engine for cruise flight or other fixed conditions. While a static simulation of the inlet shape is adequate for most circumstances, the testing requirements of military fighter craft are greater because of their high maneuverability, a feature that is accompanied by intense and brief alterations of the airflow. These transient distortions can have a significant impact on performance as well as having the potential of causing structural problems. In addition to airflow fluctuation caused by flight maneuvers, the launching of weapons into the airstream can introduce transient turbulence and exhaust gas contaminants. Currently, none of these transient effects are fully examined in wind tunnel ground tests. One solution that has been proposed to simulate this problem in gound test facilities is an array of individually controlled wedges placed in a wind tunnel ahead of the test engine. The wedges could be opened and closed, not only to simulate the given inlet shape, but also, to generate transient distortions. Coupled with a system to introduce contaminants into the flow, all manner of transient and static inlet flow distortions could be simulated in a controlled environment. This paper focuses on implementing numerical and analytical models to capture the free stream turbulence and flow distortion downstream of variable angle wedges. The numerical scheme involves application of the shear stress transport approach (using the k-ω turbulence model). The analytical approach is based on Prandtl's mixing length hypothesis and Görtler's technique for treating far wake flow behind a bluff body. The numerical and analytical solutions are compared to the experimental measurements. It is shown that while a full blown CFD simulation can aid the design of a dynamic distortion generator, a well posed analytical model may provide an expeditious alternative.

I. Introduction

JET engines go through many development and testing steps before actually being installed in an aircraft. Once a bare engine is optimized for ideal operating conditions, it is submitted to ground testing to evaluate its performance in a simulated airframe. Various methodologies are employed in wind tunnel laboratories to induce distorted flow upstream of the compressor face and, in the process, simulate a particular inlet shape. These inlet flow distortions, at present, only simulate a fixed inlet configuration over a range of flow velocities. This scenario appears to be adequate for conventional aircraft engines that will rarely, if ever, be exposed to extreme environmental and operability conditions. However, high performance systems such as those used in military fighter craft exhibit a much broader operating envelope. This is owing to fighter craft being necessarily more powerful and maneuverable than commercial systems; some of their latest generation (e.g., the Joint Strike Fighter) supplement control surfaces with vectored thrust to

*Graduate Assistant, Department of Mechanical, Aerospace and Biomedical Engineering. Member AIAA.
†Boling Chair Professor of Excellence in Propulsion, Department of Mechanical, Aerospace and Biomedical Engineering. Associate Fellow AIAA.
‡Jack D. Whitfield Professor of High Speed Flows, Department of Mechanical, Aerospace and Biomedical Engineering. Member AIAA.