IN-34

3367

p. 5

Application of Computational Fluid Dynamics in High Speed Aeropropulsion

Louis A. Povinelli
Lewis Research Center
Cleveland, Ohio

NASA

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS IN HIGH SPEED AEROPROPULSION

Louis A. Povinelli

Internal Fluid Mechanics Division
NASA Lewis Research Center
Cleveland, Ohio 44135 USA

Abstract - This paper describes the application of computational fluid dynamics to a hypersonic propulsion system, and serves as an introduction for this session. An overview of the problems associated with a propulsion system of this type is presented, highlighting the special role that CFD plays in the design.

I. INTRODUCTION

CFD has demonstrated some rather significant and spectacular achievements for aeronautical vehicles and their flow fields over the last 15 years. More recently some of these gains have been brought to bear on propulsion systems for aircraft; namely in the complex, wall bounded internal flows with energy addition inside of engines. An extensive activity has been pursued in attempting to validate numerical methods using data obtained from component (inlet, ducts, nozzles, combustors) testing. The computer codes developed have incorporated extensive physical and chemical modeling or closures, as well as utilizing multi-dimensions and sophisticated grid generation and adaptation methods. Due to the extremely complex nature of internal flow of engines, including rotating machinery, the validation and calibration of propulsion codes has proceeded slowly but steadily. With the resurgence of interest in a hypersonic air-breathing aircraft, i.e., the National Aerospace Plane, CFD efforts have been focused strongly on its proposed engine cycle. That cycle as envisioned currently relies on a supersonic combustion ramjet (Mach 6 to 15) used in conjunction with an accelerator up to Mach 6 and a rocket engine from approximately Mach 15 to orbital speed. The challenge to the scientific community is to develop accurate numerical simulations for this type of aircraft and propulsion system. Since scramjets have not been demonstrated on any propulsion system, the cycle must yet be proven feasible. In the absence of any flight data and the meager prospect of obtaining any data above flight Mach numbers of 8 in the near future, CFD becomes the tool of necessity for design of the engine and vehicle. It is worthwhile to point out, that the highly blended propulsion system makes it impossible to consider the engine without considering the influence of the airframe.

In this session, we shall concentrate on the hypersonic propulsion system and our progress is developing reliable CFD codes for analysis and design of the propulsion components. In particular, we shall look at the speed range corresponding to scramjet operation.

II. AIR CAPTURE

It should be noted at the outset that the air reaching the inlet face has experienced a rather trying time from the moment it traverses the shock wave at the aircraft nose. Depending on flight Mach number, the air may be dissociated and ionized as it moves along the underside of the aircraft. This air may undergo catalytic effects at the vehicle wall as well. Chemical and thermal non-equilibrium effects need to be modeled as well as catalicity. At the inlet plane, a substantial boundary layer has been developed on the ramp side of the inlet. It may be laminar or turbulent, or possibly transitional in nature. Shock waves from the cowl leading edge and the inlet sidewalls introduce additional complexities, such as shock-boundary layer interaction, which need to be modeled in the CFD codes. An example of such interactions is shown in a videotape. The computations, which are the result of the work of Benson and Reddy, at NASA Lewis Research Center (LeRC) illustrates a further trial for the captured airflow. The particle tracing shows that the intersection of the cowl and ramp shocks with the sidewall boundary layers causes a movement of the lower energy wall flow towards a narrow region. Cross-sectional inspection of the computed flow field reveals a vortex-like feature. At the throat, this flow behavior extends over a sufficient portion so as to cause concern regarding performance and stability of the inlet. Other rectangular inlets, such as the sidewall compression type, also experience similar effects. In this session we shall hear further discussion on inlets. Additional information is provided by this author in AGARD proceedings. Comparison of the computer inlet flow field and experimental data have shown good general aerodynamic agreement. However, in the regions where strong viscous effects are present, the agreement is marginal. Both transition and turbulence modeling improvements are required. Compressibility effects on turbulence modeling is currently being pursued as well as second moment closure by Shih and his

E-6053

cohorts at the ICOMP Center for Modeling of Turbulence and Transition at LeRC. Comparisons of heat transfer data on the inlet walls with computed results show significant differences that need to be reconciled if CFD is going to affect scramjet thermal heating design.

III. MIXING AND BURNING

Mixing and combustion of hydrogen in supersonic flow (Mach 1.5 to 7) is the critical issue to be solved for the success of scramjets. A seminal contribution to supersonic mixing was put forward by this author regarding the generation of streamwise vorticity for mixing enhancement. Current generic engine combustors rely on the concept of vorticity generation. The method of generation differs only in detail from that of this author, but not in principle. It employs swept leading edges at angle of attack to the supersonic stream to promote vorticity. Shock vortex interaction was also proposed as a means of mixing enhancement, but it is less influential than vorticity generation. The basic issue revolves about the fact that jet penetration into supersonic flows is limited to about 10 jet diameters; an amount that is insufficient for a combustion chamber. Struts protruding into the stream produce the anticipated and predictable drag, must be cooled, and must be retracted over a portion of the flight range. Some current research centers on the vorticity generation concept mentioned above using swept wall injectors with fuel injection from the back face. CFD development of three-dimensional viscous computer codes with finite rate chemistry are used to compute the mixing and reaction for these devices. Shown in a video is also an unswept configuration for comparison. The computations performed by Moon at LeRC illustrate the extent of the reacting zone for the two configurations. It should be noted that the CFD developed for the combustor does not truly represent the turbulence-chemistry interaction. The chemistry is modeled using a number of chemical steps (12) and a number of species (9). Mean values of temperature and pressure are used to determine the reaction rates. Current research is devoted toward formulating a probability density function model for the chemical reactions. Such a scheme would rely on local instantaneous values of temperature and pressure for the chemical reaction calculations. Again, we shall hear in this session, some further discussion on the mixing and combustion issue.

IV. EXHAUSTING THE AIR

The nozzle, like the inlet, blends into the aerodynamic lines of the vehicle. Here, the underside of the aft portion of the airplane forms a one-sided nozzle surface. On the opposite wall, a short cowl allows the flow to form a free shear layer with the external flow field. Hence, the nozzle dynamics and the shear layer physics and chemistry are radically different than those within our experience. Vehicle speed affects the effective back pressure on the nozzle and causes it to be over- or under-expanded. Shock-shear layer structure is dramatically affected, and can vary from shock impingement on the vehicle to no effect. The composition of the species entering the nozzle and the exit conditions influence the completeness of chemical reaction in the plume. A typical computation by Lai at LeRC using a Reynolds averaged, three-dimensional Navier-Stokes codes is shown in the video. This computation relies on a Baldwin-Lomax turbulence model. One can see the development of sidewall shear layers at the nozzle exit as well as the corresponding features on the cowl surface and shear layer. The nature of the exhaust plume is highly affected by three dimensionality. Only limited data exist for flow field comparison at the present time. There is no doubt, however, that significant closure issues remain to be addressed.

V. CONCLUDING REMARKS

On an overall basis, one can observe that a significant amount of progress has been made on the application of CFD for high-speed airbreathing propulsion systems. Excellent qualitative agreement is the usual picture, with significant discrepancies only in those near wall regions dominated by strong viscous flows. Nonequilibrium air effects and finite rate chemistry are extensively modeled and computed. However, proper turbulence chemistry interaction requires a significant amount of attention. Improvements in turbulence and transition models are also critically needed. I look forward to hearing the presentations in this session on these important issues; I hope you share my enthusiasm.

2

NASA

National Aeronautics and Space Administration

Report Documentation Page

1. Report No. NASA TM – 103780	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Application of Computational Fluid Dynamics in High Speed Aeropropulsion		5. Report Date
		6. Performing Organization Code
7. Author(s) Louis A. Povinelli		8. Performing Organization Report No. E – 6053
		10. Work Unit No. 505 – 62 – 52
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135 – 3191		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546 – 0001		14. Sponsoring Agency Code

16. Abstract

This paper describes the application of computational fluid dynamics to a hypersonic propulsion system, and serves as an introduction for this session. An overview of the problems associated with a propulsion system of this type is presented, highlighting the special role that CFD plays in the design.

17. Key Words (Suggested by Author(s)) Computational fluid dynamics Hypersonics Propulsion High speed	18. Distribution Statement Unclassified - Unlimited Subject Category 34

19. Security Classif. (of the report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of pages 3	22. Price* A02