IN-34
/684
p 17

CFD for Hypersonic Propulsion

Louis A. Povinelli
Lewis Research Center
Cleveland, Ohio

NASA

CFD FOR HYPERSONIC PROPULSION

Louis A. Povinelli

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

Introduction

This paper presents some research activity on the application of CFD for hypersonic propulsion systems, which the author presented at the workshop. Since this author was requested, on one days notice, to fill in for an invited speaker who was unable to attend, the material contained herein is more representative of what was in hand rather than the total CFD effort that is underway at the authors laboratory and elsewhere. The presentation addressed the following items:

- o Propulsion system integration
- o Typical computations for propulsion components
- o CFD validation issues
- o Prognoses for success

To a large extent, the comments and illustrations used herein are based on a presentation made by the author at the Ninth ISABE meeting in Greece in September 1989 (ref. 1).

Propulsion System Integration

In the mid 1970's, propulsion testing of a hypersonic ramjet engine (HRE) was performed at Mach 5 to 7 at the Plum Brook Station of the NASA Lewis Research Center. That configuration was axisymmetric in design and had a rather small annular passage through the combustor. The HRE was representative of a pod-mounted system rather than the highly blended configurations of today. In this presentation, the ability of CFD codes to simulate propulsion system components is discussed relative to the integrated engine body configurations which are more typical of today's designs. A generic version of such a highly blended configuration is shown in figure 1.
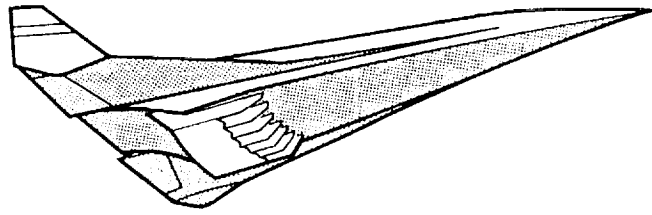
Figure 1.—Blended wing body configuration.

Forebody compression is considered an essential feature of such a design and the nature of the corresponding boundary layer must be taken into consideration for inlet behavior. Needless to say, a great deal of vehicle research and testing is required for proper aerodynamics as well as being capable of delivering uniform flow to the inlet. The propulsion system is assumed to be a combined ramjet/scramjet system having a common flowpath.

Propulsion Modes: The common flowpath engine considered in this paper is envisioned to operate as a subsonic combustion ramjet over the flight Mach number from 3 to 6. At higher flight speeds, the supersonic combustor mode would be employed up to flight numbers which may be on the order of Mach 15. This upper limit is speculative and depends on a number of unknown factors. Above the upper limit of air-breathing operation, integrated rocket thrusters would be employed to achieve orbital velocities. The ramjet modes are illustrated in figure 2.

Hypersonic Propulsion Design Approach: Given the aircraft propulsion system illustrated in figure 2 and the operational modes for the engine, one may inquire how to approach its propulsion design.
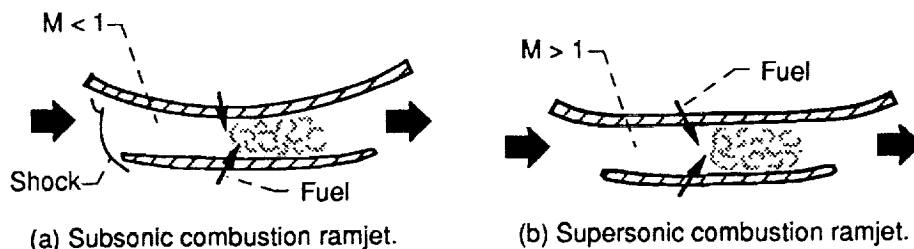

(a) Subsonic combustion ramjet.   (b) Supersonic combustion ramjet.

Figure 2.—Ramjet operating modes.

2

The current philosophy runs as follows; existing computer codes with the "best" turbulence and chemistry modeling are assessed against the existing data base which is mostly at Mach numbers less than 8. Where a lack of data exists, then new experiments must be performed. Numerous iterations between computations and experiments will eventually "validate" the codes. These validated codes, with all the sophistication of real gas effects and turbulence/chemistry closures will subsequently be extrapolated to the higher Mach numbers (e.g., M8 to M16) to assess various geometrical engine configurations. After a "sufficient" number of numerical computations, backed up by available pulse or shock facility data, a candidate design will emerge.

Flight experiments will then provide the next or "true" level of validation. Information from such testing will then be used to modify the physical and chemical modeling used in the simulations. As flight test speeds are increased incrementally over the Mach range required for orbit, the improved CFD simulations will provide guidance at each of the next incremental speed levels. Thus, flight testing and CFD simulations will be conducted "hand-in-hand" as hypersonic vehicles move up the speed corridor.

Above the upper limit of air-breathing operation, integrated rocket thrusters would be employed to achieve orbital velocities.

Typical Computations for System Components
Generic Inlet

The simple rectangular inlet configuration shown in figure 3 was tested at Mach 12.26. A flat plate of 30 in. length preceded the entrance to the inlet in order to simulate the boundary layer growth on the forebody of a hypersonic aircraft. Compression wedges form the top and bottom walls of the inlet and the contraction ratio was equal to five. Swept sidewalls which connect the

3

57.5 in.

51 in.

48 in.

Cowl

Ramp

Flat plate
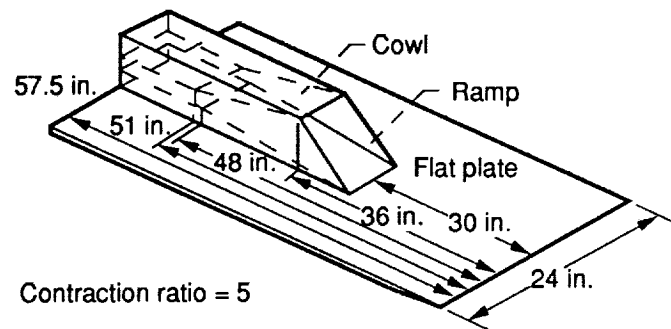
36 in.   30 in.

24 in.

Contraction ratio = 5

Figure 3. - Generic inlet (ref. 5).

upper and lower walls, prevent compressed flow from
spilling over the inlet sidewalls.

Computations were made with a three-dimensional PNS
LBI implicit scheme (ref. 2) with grids of 80 by 60 by
750 on a Cray X-MP. This solver includes real gas
effects (ref. 3) as well as dissociation and ionization
modeling (ref. 4). For this experiment, however, the
inlet air was only heated sufficiently to avoid conden-
sation, and the real gas modeling was not required. The
issues that are of importance in this computation are the
assumptions regarding the state of the boundary layer,
the turbulence model, spillage of flow around the side-
plates and shock boundary layer interaction. For the PNS
computation it was assumed that the boundary layer was
turbulent starting on the leading edge of the flat plate,
the cowl leading edge and the sidewall leading edges.
The turbulence model used was a Baldwin-Lomax model and
spillage was not considered. Modeling of the shock
boundary layer interaction involved the use of a flare
approximation in order to allow the PNS to march through
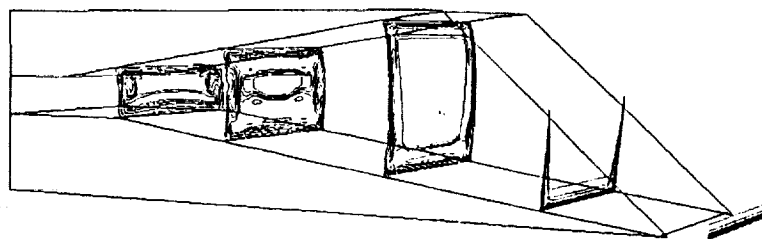the region of flow separation. The results of the PNS

Figure 4.—Mach number contours, M = 12.25 (ref. 2).

4

solution are shown in figure 4. Contour plots of con-
stant Mach number within the inlet are shown. The con-
centration of lines near the walls indicate the boundary
layers, while concentrated contours in the freestream,
indicate shock wave locations. The flow features seen
are boundary layer buildup on the flat plate followed by
thickening on the sidewalls and ramp surface. Shocks
generated by the compression wedges are seen as horizon-
tal lines, and the sidewall shocks are vertical lines.

The low energy flow in the sidewall boundary layer
has been swept up the sidewall by the ramp shock, and
then down the sidewall by the cowl shock. Further down-
stream, the shock waves cross and are distorted by
interaction with the sidewall boundary layers and the
expansion fan on the ramp surface. Additional complex
interactions then occur as the flow moves downstream.
The PNS solution fails when the ramp shock wave reflects
from the cowl and strikes the ramp surface, resulting in
large corner separation of the low energy flow.

An alternate view of the three-dimensional flow is
obtained with sidewall particle tracing (fig. 5). Inter-
action of the ramp and cowl shocks with the sidewall
boundary layer causes the particles to converge near the
shock interaction point. The particles are then dis-
placed due to the vortex motion. Flow migration details
are evident in this computational simulation. As a side-
note, since the vortex persists downstream, it has been
proposed that enhanced fuel mixing could occur with judi-
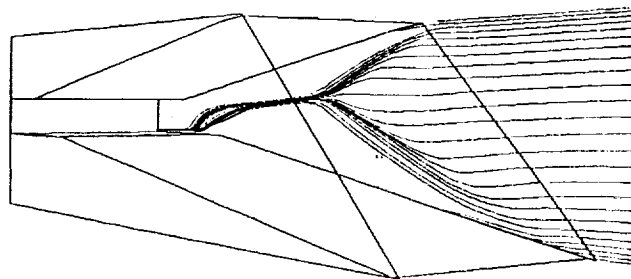cious injector locations downstream (ref. 1).

Figure 5.—Sidewall particle tracing, M = 12.25 (ref. 2).

5

Navier-Stokes computations have also been carried
out for the generic inlet at NASA Langley with CFL3D
(ref. 5). In this case, the boundary layers were assumed
turbulent on all surfaces from the leading edges. The
turbulence model used was a Baldwin-Lomax model and
spillage over the sideplates was not considered. In the
vicinity of the shock boundary no special modeling was
employed. Figure 6(a) shows the pressure distributions
for the ramp and centerline cowl surfaces, using two
different grids. Figure 6(b) shows the side plane dis-
tributions. Comparison of the CFL3D results and the
experimental data show good agreement, particularly along
the centerline where shock locations appear to be well

(a) Ramp and cowl centerlines.

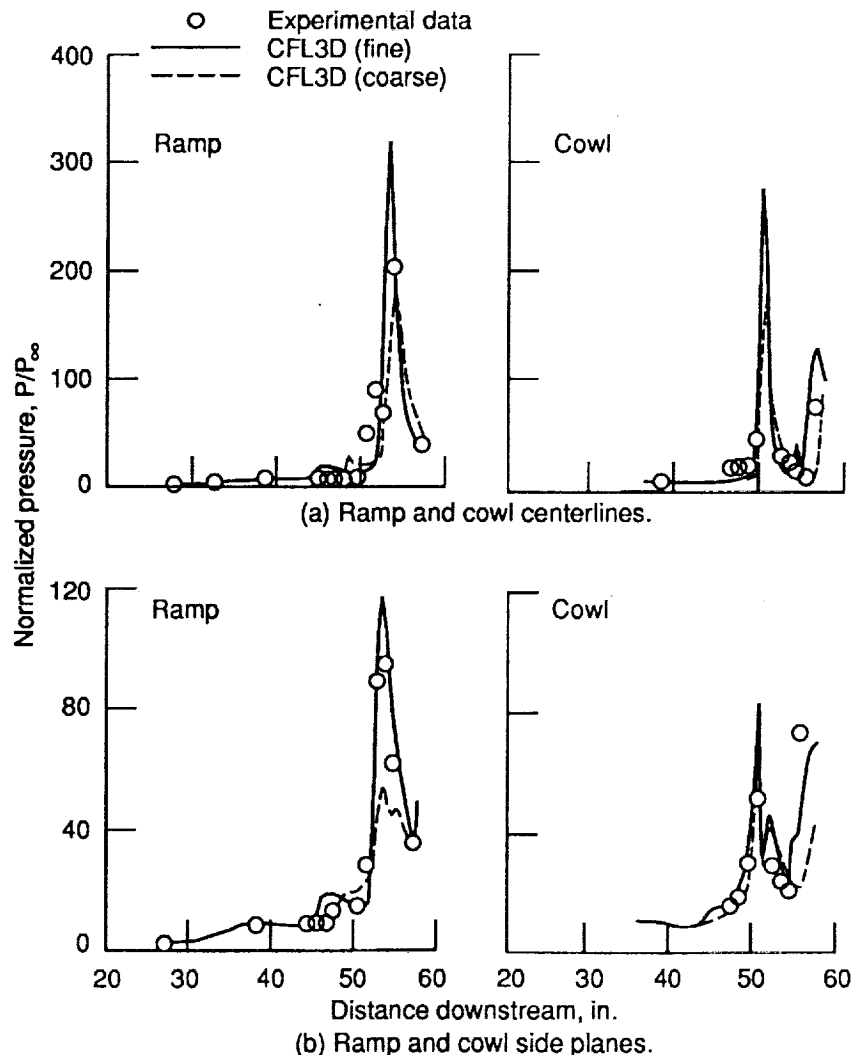(b) Ramp and cowl side planes.

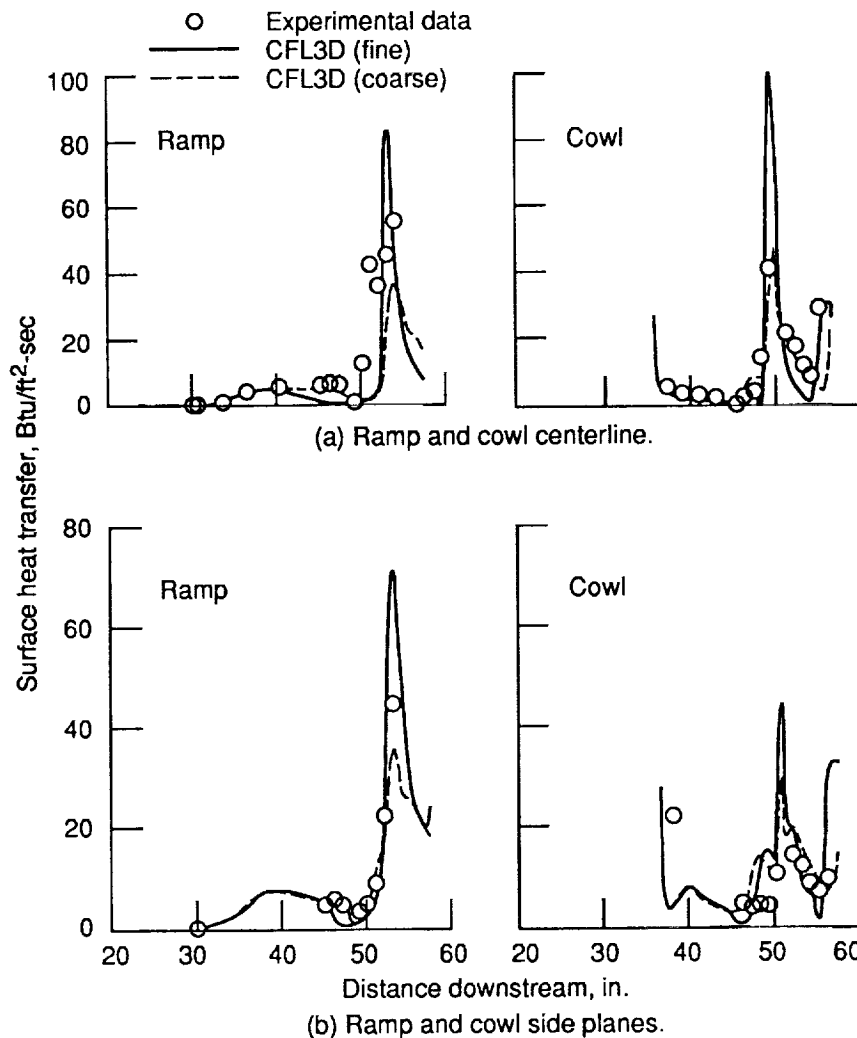Figure 6.—Pressure comparisons between CFL3D and experiment (ref.5).

Figure 7.—Heat transfer comparisons between CFL3D and experiment-2-D model, ramp side plane and cowl side plane (ref.5).

resolved by the code. The viscous interactions occurring along the side plane are also accurately resolved. In addition, CFL3D was used to compute the heat transfer on the ramp and cowl surfaces (figs. 7(a) and (b)). The heat flux distributions are reasonably well predicted on both ramp and cowl surfaces.

Strong viscous effects are evident along the side walls of the inlet in agreement with the complex behavior shown in figures 4 and 5. Further analysis of the Mach 12 inlet is underway at the NASA Centers and industry.

7

Combustors

Simple Combustors: The simplest supersonic combus-
tor scheme is a channel with a single jet of hydrogen
injected normally to the supersonic stream, as illus-
trated in figure 8.

M = 4

T = 1300 °K
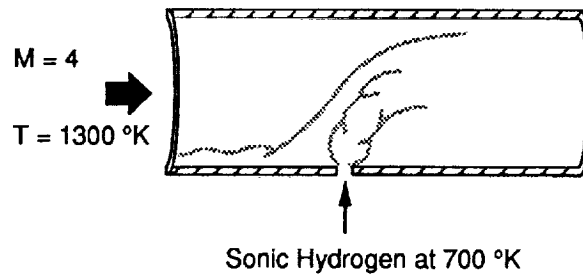
Sonic Hydrogen at 700 °K

Figure 8.—Simple jet injection.

This reacting flow situation was solved using the
RPLUS code at NASA Lewis, which is an LU algorithm. The
grid used for the solution was 60X40X40 with grid clus-
tering. The resulting Mach number distribution is shown
in figure 9 with good fidelity and resolution of the

3.8

3.8

3.8

3.8

3.8

3.8

1.1

1.4

3.8

3.2

3.2

3.2

x = 0.615 cm

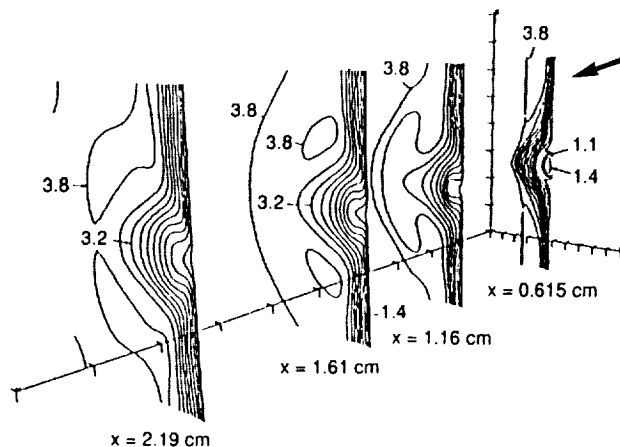1.4

x = 1.16 cm

x = 1.61 cm

x = 2.19 cm

Figure 9.—Mach number contours on yz planes at
various x locations for Case 1 (ref. 6).

injection fluid mechanics. Figure 10 shows the computed
temperature contours by Drs. Yu and Shuen (ref. 6).

Dual Injection: A somewhat more complex injection
scheme involves two jet injection ports which are aligned
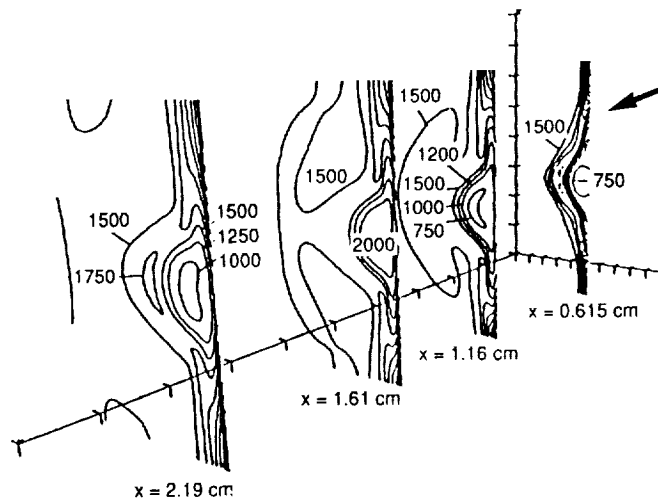in the axial direction as illustrated in figure 11.

8

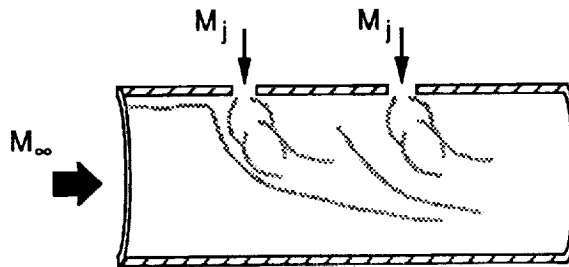Figure 10.—Temperature contours on yz planes at various x locations for Case 1 (ref. 6).
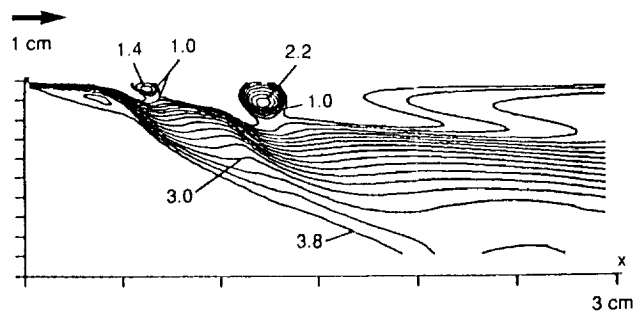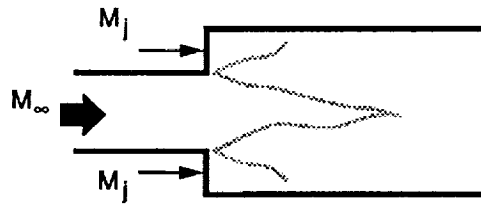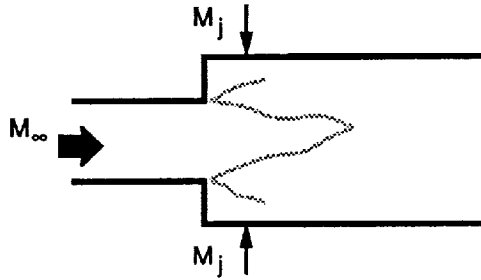

Figure 11.—Dual jet injection.

Figure 12.—Mach number contour on xy plane at center of injection port for Case 2 (ref. 6).

The resulting Mach number distribution from the RPLUS code by Yu and Shuen (ref. 6) is shown in figure 12. Both Mach disc structures are discernable in the computations.

9

(a) Parallel injection.

(b) Transverse injection.

Figure 13.—Sudden expansion combustor.

(a) Parallel injection.
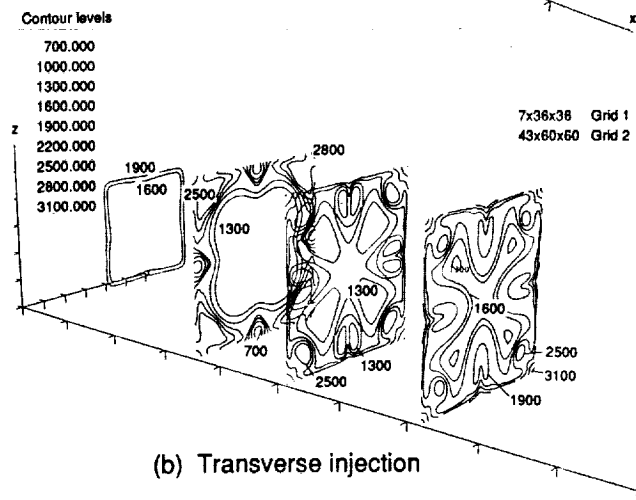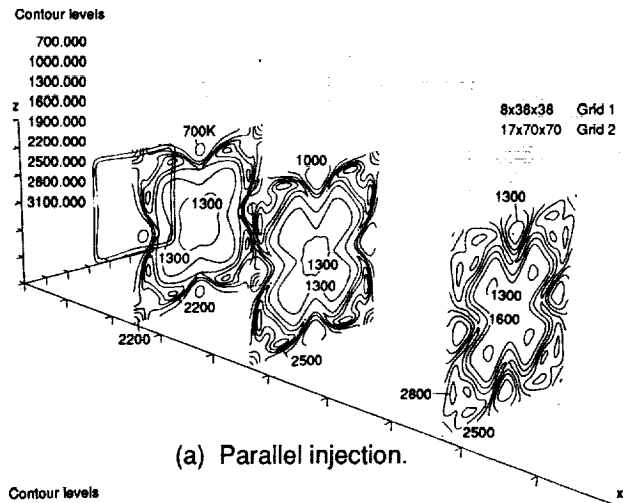

(b) Transverse injection

Figure 14.—RPLUS temperature computations for a sudden expansion combustor by Tsai (ref. 7).

Additional complexity is introduced by modifying the straight walls of the combustor so that a sudden expansion or a rearward facing step results, as shown in figure 13.

Computations using the three-dimensional RPLUS code with hydrogen-air chemistry by Dr. Tsai (ref. 7) is shown in figures 14(a) and (b). The temperature distributions appear reasonable. The results shown are laminar. Turbulence modeling needs to be incorporated into RPLUS for more realistic conditions. That activity is currently nearing completion.

Asymmetric Nozzle Geometry

A typical three-dimensional Navier-Stokes computation for a nonaxisymmetric nozzle is illustrated in figure 15. For this case, the supersonic jet is issuing
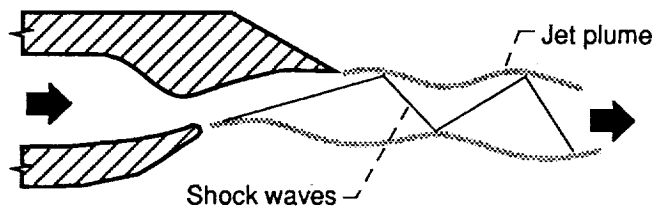

Figure 15.—Asymmetric nozzle configuration.

into a quiescent atmosphere. A three-dimensional Navier-Stokes code, PARC, was used to study the flow behavior. A Baldwin-Lomax turbulence model was employed in the code. These computations were performed by Dr. Hen Lai of Sverdrup/Lewis (ref. 8). Figure 16 shows typical results of the spanwise variation of Mach number, starting at the nozzle center plane and extending to the side wall shear layers. Analyses of the type described above have been combined to provide a complete computation from vehicle nose through the propulsion system to the tail of the aircraft.
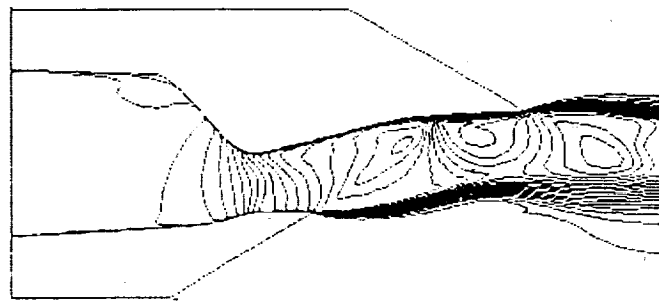
(a) Centerline.
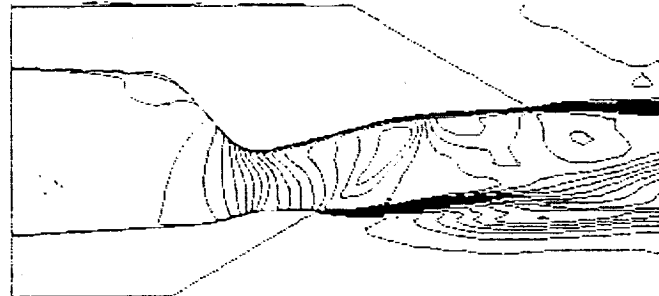
(b) Midpoint.
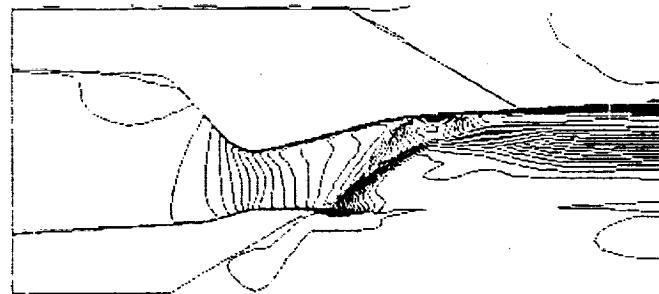
(c) Wall.

Figure 16.—Mach number contours, side views (ref. 8).

Validation Issues

Validation of numerical simulations must deal with a number of specific issues. In this section we will address:

o General validation issues

o Design issues

o Critical research fora validation

General Issues: In this category, both computational and experimental issues need to be addressed. On the computational side, modeling of turbulence, boundary layer transition and reaction chemistry is of paramount importance. Sensitivity to internal code parameters,

grid sensitivity and the effect of numerical boundary
conditions must also be ascertained in the process of
establishing code validity. In addition, convergence
behavior; the ability to capture discontinuities and to
preserve mass, momentum, energy and species must be dem-
onstrated. Corresponding issues on the experimental side
revolve around the validity, repeatability and accuracy
of data. The initial conditions must be documented to a
greater extent than has been usually done in the past.
In addition, some testing is strongly affected by the
experimental apparatus itself, such as wind tunnel walls,
so that numerical simulations are only meaningful for the
entire apparatus. At any rate, the effect of the flow
facility must be known and measured. Finally, the need
for fluctuating, nonintrusive data remains an important
requirement.

Design Issues: As computer codes are put to use for
"design" purposes, a number of critical questions arise.
Some typical questions that have arisen are the
following:

- o To what extent are propulsion CFD codes
  validated?
- o What is the degree of validation required?
- o What are the propulsion design needs?
- o What engineering parameters are needed?
- o What computed variables are required to produce
  engineering parameters?
- o To what extent are the computed variables affec-
  ted by specific physical or chemical modeling?

Many discussions have ensured over the above ques-
tions, without a clear cut response to all of the inquir-
ies. In general, one is led to the conclusion that a
systematic study and resolution of all the computational
and experimental validation issues raised in the previous
section would require many years to complete. It appears
prudent, therefore, at this time to develop fundamental
understanding first, then to make judgements on the

13

importance of various phenomena and lastly to perform numerical sensitivity studies.

Prognosis for Success

It is believed that modeling of the most important physics and chemistry will be improved only through a rigorous and systematic validation process. This process will involve a large number of experiments, from those with simple isolated phenomena to those involving multiple simultaneous effects. Corresponding modeling and code simulations must also be carried out in close concert with the measurements.

Furthermore, an improved understanding of the relevant flow physics and chemistry over a wide range of operating conditions will evolve incrementally from experimental flight data. These data will assist in the improvement of the required modeling for CFD codes and the codes, in turn, can be applied to the next increment in the vehicle's flight velocity.

Finally, it is believed that validated (which implies an important role of experiments) numerical simulations will play a major role in the design of future propulsion systems only if continuous effort is focused on the combined experimental/computational methodology.

References

1. Povinelli, L.A.: Advanced Computational Techniques for Hypersonic Propulsion. International Symposium on Air Breathing Engines (9th, 1989, Athens, Greece) F. Billing, ed., AIAA, Washington, DC, 1989, Vol. 2, pp. 993-1008. (Also, NASA TM-102005, 1989.)

2. Reddy, D.R., et al.: Three-Dimensional Viscous Analysis of a Hypersonic Inlet. AIAA Paper 89-0004, Jan. 1989. (Also, NASA TM-101474, 1989.)

3. Liou, M.-F.: Three-Dimensional PNS Solutions of Hypersonic Internal Flows With Equilibrium Chemistry. AIAA Paper 89,0002, Jan. 1989.

4. Yu, S.-T., et al.: Numerical Simulation of Hypersonic Inlet Flows With Equilibrium or Finite Rate Chemistry, AIAA Paper 88-0273, Jan. 1988.

5. Diley, A.; Eppart, W.; and Switzer, G.: Zonal Analysis of Two High Speed Inlets. Proceedings of the CFD Symposium on Aeropropulsion, NASA CP-10045, 1990, pp. 29-1 to 29-13.

6. Yu, S.-T.; Tsai, P.; and Shuen, J.-S.: Three-Dimensional Calculation of Subsonic Reacting Flows Using an LV Scheme. AIAA Paper 89-0391, Jan. 1989.

7. Lai, H.: Three-Dimensional of Single-Expansion-Ramp and Scramjet Nozzles. Proceedings of the CFD Symposium on Aeropropulsion, NASA CP-10045, 1990, pp. 32-1 to 32-24.

8. Tsai, Y.L.: Recent Update of the RPLUS2D/3D Codes. Proceedings of the CFD Symposium on Aeropropulsion, NASA CP-10045, 1990, pp. 34-1 to 34-12.

NASA
National Aeronautics and
Space Administration

Report Documentation Page

1. Report No. NASA TM-103791	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle CFD for Hypersonic Propulsion		5. Report Date
		6. Performing Organization Code
7. Author(s) Louis A. Povinelli		8. Performing Organization Report No. E-5689
		10. Work Unit No. 506-62-21
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 presents an overview of research activity on the application of CFD for hypersonic propulsion systems. After an initial consideration of the highly integrated nature of air-breathing hypersonic engines and airframe, our attention is directed toward typical computations carried out for the components of the engine. A generic inlet configuration is considered in order to demonstrate the highly three-dimensional viscous flow behavior occurring within rectangular inlets. Reacting flow computations for simple jet injection as well as for more complex combustion chambers are then discussed in order to show the capability of viscous finite rate chemical reaction computer simulations. Finally, nozzle flow fields are demonstrated, showing the existence of complex shear layers and shock structure in the exhaust plume. In concluding, we examine the general issues associated with code validation as well as the specific issue associated with the use of CFD for design. A prognosis for the success of CFD in the design of future propulsion systems is offered.

17. Key Words (Suggested by Author(s)) CFD Hypersonic propulsion Numerical methods Propulsion CFD	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 16	22. Price* A03