



AIAA 92-0839

**NAVIER-STOKES SIMULATION OF
PLUME/VERTICAL LAUNCHING SYSTEM
INTERACTION FLOWFIELDS**

**B.J. York, N. Sinha, and S.M. Dash
Science Applications International Corporation
Propulsion Fluid Dynamics Division
Fort Washington, Pennsylvania**

**L. Anderson and L. Gominho
Naval Surface Warfare Center
Dahlgren, Virginia**

**30th Aerospace Sciences
Meeting & Exhibit
January 6-9, 1992 / Reno, NV**

Report Documentation Page				Form Approved OMB No. 0704-0188	
Public reporting burden for the collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to a penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.					
1. REPORT DATE JAN 1999		2. REPORT TYPE		3. DATES COVERED 00-01-1999 to 00-01-1999	
4. TITLE AND SUBTITLE Navier-Stokes Simulation of Plume/Vertical Launching System Interaction Flowfields				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S)				5d. PROJECT NUMBER	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NAVSYS Corporation,14960 Woodcarver Road,Colorado Springs,CO,80921				8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)				10. SPONSOR/MONITOR'S ACRONYM(S)	
				11. SPONSOR/MONITOR'S REPORT NUMBER(S)	
12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution unlimited					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT					
15. SUBJECT TERMS					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES 12	19a. NAME OF RESPONSIBLE PERSON
a. REPORT unclassified	b. ABSTRACT unclassified	c. THIS PAGE unclassified			

NAVIER-STOKES SIMULATION OF PLUME/VERTICAL LAUNCHING SYSTEM INTERACTION FLOWFIELDS*

B.J. York,† N. Sinha,†† and S.M. Dash†††
Science Applications International Corporation
Fort Washington, Pennsylvania

L. Anderson** and L. Gominho***
Naval Surface Warfare Center
Dahlgren, Virginia

Abstract

The application of Navier-Stokes methodology to the analysis of Vertical Launching System/missile exhaust plume interactions is discussed. The complex three-dimensional flowfields related to the Vertical Launching System (VLS) are computed utilizing the PARCH/RNP Navier-Stokes code. PARCH/RNP solves the fully-coupled system of fluid, two-equation turbulence ($k\epsilon$) and chemical species equations via the implicit, approximately factored, Beam-Warming algorithm utilizing a block-tridiagonal inversion procedure.

Introduction

The PARCH Navier-Stokes code has been under continual development at SAIC for the past several years. PARCH was originally developed as an outgrowth of the NASA/Ames ARC¹ and AEDC PARCH^{2,3} Navier-Stokes codes, and has been upgraded with respect to turbulence modeling, finite-rate chemistry, and nonequilibrium particulate modeling. The PARCH code framework provides specialized grid blanking capabilities which permit treating complex propulsive geometries using relatively straightforward grids. The methodology in PARCH for incorporating a two-equation $k\epsilon$ turbulence model, matrix-split finite-rate chemistry, and nonequilibrium particulates has been described in previous papers (see Refs. 4-13). The application of PARCH to a variety of flowfield

problems under various programs has resulted in the development of several specialized versions as shown in Table I. Depending on the flowfield to be simulated, these various versions contain different turbulence models, chemistry and/or particulate models, as well as different algorithms and boundary conditions. The rationale for specialized versions rather than having a single version applicable to many different problems is discussed in References 13 and 14.

Current work is proceeding towards consolidating appropriate features of these several versions into a new version, PARCH/RNP, which will emphasize the simulation of missile plume/airframe/launcher interaction flowfields. The features to be contained in PARCH/RNP are summarized in Table II. Unlike the earlier rocket nozzle (RN) and tactical missile (TMP) versions, which emphasized matrix-split/loosely-coupled chemistry and turbulence modeling methodology and the diagonalized matrix inversion procedure, the new RNP version will employ large-matrix/strongly-coupled methodology and more robust block matrix inversions.

The requirement to develop a new RNP version of PARCH has resulted from assorted numerical studies of complex plume flowfields where the other versions had deficiencies in obtaining a converged solution. The TMP and RN versions both emphasized the analysis of predominantly inviscid flows with wall-bounded viscous flow zones, which are relatively stable. The diagonalization technique which was originally employed requires an explicit treatment of viscous terms and precludes a time-accurate analysis. This algorithm is catered to taking independent local time-steps to accelerate convergence to steady-state, and employs matrix-splitting for nonequilibrium terms rather than catering to the smallest time-scales of the problem which would require taking very small time-steps (see Ref. 14). This methodology has worked extremely well for rocket nozzle flows and for tactical missile aerodynamic flows with nozzle exhaust plume interactions.

Such methodology has not been successful for analysis of less stable free shear flows such as plumes

* This work was supported by the Naval Surface Warfare Center in Dahlgren, Virginia, through Contract No. N60921-90-C-A366.

† Senior Research Scientist, Member AIAA

†† Technical Director, Member AIAA

††† Chief Scientist and Asst. VP, Member AIAA

** Senior Mechanical Engineer

*** Mechanical Engineer

Table I. OPERATIONAL VERSIONS OF PARCH NS CODE

CODE NAME	PARCH/RN	PARCH/GTP	PARCH/LF	PARCH/TMP
APPLICATION	rocket nozzle	gas turbine nozzle/plume	chemical laser	tactical missile aero/plume
SPONSOR	SDIO/MICOM	AFEWC/NASA-LaRC	MICOM	MICOM
EQUATIONS	Axi	2D/Axi/Swirl	2D and 3D	2D/Axi and 3D
CHEMISTRY	finite-rate	two-stream one-step global reaction	finite-rate	finite rate, equilibrium air, two-stream, perfect gas
TURBULENCE	algebraic	$k\epsilon$	none	$k\epsilon$ /Chien
BLANKING	none	generalized	generalized	generalized
PARTICULATES	nonequilibrium	none	none	equilibrated
ALGORITHM	Beam-Warming, diagonalized fluids (4 x 4), matrix-split chemistry, spatial/implicit particle solver	Beam-Warming, strongly-coupled fluids/species/turbulence - 8 x 8 block inversion	Beam-Warming, strongly-coupled fluids/species 4 + n or 5 + n block size	Beam-Warming, diagonalized fluids (4 x 4 or 5 x 5), matrix-split chemistry, weakly-coupled turbulence

Table II. PARCH/RNP 2D/AXI AND 3D NS CODES

PARCH/RNP 2D/AXI AND 3D NS CODES	
APPLICATION	rocket nozzles and plumes, missile airframe/launcher interactions
SPONSOR	MICOM
EQUATIONS	2D/Axi/Swirl-Spin and 3D
CHEMISTRY	finite-rate, equilibrium air, two-stream, one-step global reaction, perfect gas
TURBULENCE	$k\epsilon$ /Chien with compressibility/swirl corrections
PARTICULATES	equilibrated mixture and generalized nonequilibrium (under development)
BLANKING	generalized 2D/3D and multi-zone blocking (under development)
ALGORITHM	strongly-coupled fluid/species/turbulence, 5 + NS + 2 block matrix inversion
NEW WORK	extensions of particulate methodology to include particle/particle interactions, volumetric effects, particle/turbulence interactions

exhausting into a quiescent or low-speed external stream. It has also been problematic for viscous dominated problems (e.g., the low Re flow in laser cavities) and most problems with free shear layers where one (or both) of the streams is subsonic and disturbances can propagate upstream (e.g., core/fan mixing in a gas turbine). For such problems, the additional stability/robustness provided by a fully-implicit/strongly-coupled approach is required, and the path to convergence may require the use of time-accurate methodology to achieve a steady state solution. While PARCH/RNP will provide such capabilities, it is not intended as a replacement for the earlier TMP and RN codes – only as an alternate to be used where required. In the more stable environment encountered in strategic missile flowfields, for example, where high Mach number flight conditions prevail, the diagonalized/matrix-split approach would prove to be significantly more efficient.

This paper will describe the application of PARCH to VLS related flowfields. The VLS is a ship-based missile launcher as schematized in Figure 1. The launcher consists of several launch canisters containing missiles which are installed below the ship's deck. When a missile is fired, its exhaust gases empty into the plenum chamber under the launch canisters and are redirected up through the uptake exhaust duct. Analysis of this problem is being performed by separately considering the internal duct flowfield and the external plume flow of the uptake exhaust. Knowledge of the external uptake plume flowfield is important since the missile flies through this plume. The details of the internal plenum/uptake flowfield are of interest since the hot particle-laden exhaust products can cause high pressure loads and severe heating rates to the system.

Previous calculations of VLS related flows have been performed with either one-dimensional gas dynamics codes or two-dimensional hydro-codes. Although these codes have provided many useful results, it has long been recognized that in general, the flow inside the VLS is highly three-dimensional. This fact has been dramatized by results from recent land based tests. These results show that the erosion of thermal protection materials, particularly in the lower regions of the uptake are highly irregular. Furthermore, the erosion patterns appear to be affected by the position of the active missile, (e.g., a middle or end canister) and the missile load out geometry. The introduction of multi-nozzle missiles into the VLS means that the rocket motor plume both inside the missile canister and above the deck can no longer be approximated by two-dimensional calculations. The internal flow in the plenum and the uptake is further complicated by the presence of aluminum oxide

particles of various sizes, viscously separated flow regions and chemical reactions between the exhaust products and the air initially in the system. The PARCH codes described above have promise of becoming valuable tools for calculating VLS related flows.

Results

The following describes preliminary experiences in the use of PARCH to calculate flows of interest to the VLS. Three types of flows were calculated: A plume from a four nozzle missile exhausting into the atmosphere, the flow in a simplified VLS plenum/uptake geometry, and the flowfield resulting from an uptake plume in the presence of a 60 knot relative head wind. The results of these calculations are described below.

Uptake Plume Simulation

The exhaust plume which issues from the uptake is essentially a rectangular sonic underexpanded jet exhausting into quiescent air (or a small cross-flow if the ship is moving or there is wind). Since the missile may fly through this plume it is important to know the effects of the plume induced loading on the missile. The flow conditions for the uptake plume calculation performed are as follows: (The freestream represents a 60 knot cross wind.)

<u>JET</u>		<u>FREESTREAM</u>	
P_j	= 1.66 atm	P_∞	= 1 atm
T_j	= 3470 K	T_∞	= 236 K
M_j	= 1	M_∞	= .1

Initial attempts at analyzing this uptake plume flowfield were made at the 2D level using the diagonalized/loosely-coupled version of PARCH. This flowfield consisted of a sonic, underexpanded ($P_j/P_\infty = 1.66$), hot jet exhausting into a quiescent environment. Calculations did not yield a converged solution. Figure 2 presents contours of total enthalpy illustrating the instabilities which were encountered. This flowfield was recomputed with a preliminary version of PARCH/RNP with the fully-coupled equation set and block matrix inversion which yielded a stable converged solution. Figure 3 exhibits contours of total enthalpy for the converged solution.

For the three-dimensional uptake problem, a grid of 121 (x), 31 (y), and 71 (z) was employed with half plane symmetry about the $Y = 0$ plane. The exhaust slot is approximately 8 inches wide by 100 inches long. Initial attempts to compute a steady, turbulent 3D jet

were unsuccessful. Figure 4 shows temperature contours in the X-Z symmetry plane illustrating the instabilities that have been encountered. Various attempts were made at initializing the jet along different trajectories to begin the calculations. However, all the calculations become unstable after a short time. Another approach was taken using the time-accurate run option in PARCH/RNP. The computational domain was initialized with the freestream conditions, and the jet conditions were applied only on the jet boundary plane at $t = 0$. The calculation then proceeded in a time-accurate manner allowing the jet to penetrate into the freestream flowfield. Figure 5 shows Mach number contours in the X-Z symmetry plane at several stages in this calculation. Note that the jet remains stable in the nearfield region (up to 15 jet widths) after which it begins to break up. This may explain why the earlier attempts at computing this as a steady flowfield were unsuccessful.

Plenum/Uptake Simulation

The VLS plenum/uptake is another problem area being analyzed. A sketch of the configuration is illustrated in Figure 6. The flowfield in this region is very complex, in that the exhaust from the missile being fired fills the plenum and then exhausts upward through the uptake. The exhaust is very energetic and contains particulates from the solid rocket propellant. The erosion of the floor of the plenum and various other surfaces is of major concern. The analysis of this problem is being performed by making simplifying approximations to the geometry and flowfield, and gradually introducing various complexities into the analysis. Figure 7 shows the results of a 2D calculation¹⁵ illustrating the primary features of the flowfield. Three dimensional calculations have recently been initiated on the simplified geometry shown in Figure 8. This configuration contains major features of the flow but omits the various internal structures in the actual plenum. The initial analysis considers exhaust products as a perfect gas for the missile exhaust and no attempt is made to resolve the wall boundary layers. Note that the ability to readily grid such complex configurations is expedited by the specialized blanking logic available from PARC.^{2,3} Figure 9 shows the Mach number variation in the symmetry plane in the center of the plenum/uptake duct. The flow has not yet become choked at the exit of the uptake. Figure 10 shows Mach number contours in the symmetry plane. The nozzle exhaust shear layers are evident, as well as an impingement shock near the plenum floor due to the exhaust flow.

Multi-Nozzle Plume Simulation

Benchmark calculations for the measured free field plume of a simulated four nozzle rocket motor configuration were performed with the PARCH code. The experiment consisted of four cylindrical nozzles with radii equal to .276 inches protruding from a cylindrical base region with diameter equal to 2 inches, as schematized in Figure 11. The gas utilized was air at Mach 2.6, $P = 4.46$ atm, and $T = 224^\circ\text{K}$, exhausting into a quiescent environment. Perfect gas thermochemistry was employed. To simplify grid generation requirements, the nozzles were taken to be flush with the missile base. The grid utilized contained $121 \times 31 \times 71$ points, assuming $\frac{1}{8}$ plane symmetry. Figure 11 illustrates the grid in the X-Z symmetry plane and the inflow plane, respectively. Turbulence was simulated with the two-equation $k\epsilon$ turbulence model. Figure 12 shows the computed density contours in the X-Z symmetry plane illustrating the complex wave structure and wave/shear layer interactions within the jets. Figure 13 shows a comparison between the experimental and computed pitot pressure profiles at several axial locations in the X-Z symmetry plane. The results agree reasonably well, with the greatest discrepancy being on the centerline. It should be noted that our simplified treatment of the nozzle geometry could have an impact on the computed jet flowfield and the comparison with the experimental data since the actual nozzle protrusion was not negligible. Also, the protruding nozzles have a finite base thickness (as depicted by the outer rings in Fig. 11). These geometrical details will have an effect on the computed flow structure and may be a source of the discrepancies in the comparison of the computed and measured pitot pressure values.

Concluding Remarks

Progress towards simulating a variety of complex VLS related missile plume/airframe/launcher flowfields using versions of the PARCH NS code has been described in this paper. Achieving stable, converged solutions is highly problem dependent and different numerical procedures and convergence strategies are required for different problems. The most difficult problems are those entailing the simulation of plumes exhausting into a quiescent or subsonic external stream which can be extremely unstable. For such flows, strongly-coupled numerics and time-accurate convergence strategies are required. Experience indicates that it is not feasible to have a single generalized code which contains all the numerical algorithms, turbulence models, and chemistry variants for applications to all classes of fluid dynamic problems. This has resulted in the development of

several specialized versions of PARCH for different applications. Based on this experience, the latest version (PARCH/RNP) will contain the features which are most suitable for efficient solution of tactical missile airframe/plume/launcher interaction flowfields.

Although some numerical difficulties were encountered in the calculations described above, the applications of PARCH, particularly the PARCH/RNP code for VLS related flows is encouraging. Future studies will focus on modifying the code to include such features as non-equilibrium particle gas flows in internal VLS environments, time accurate finite rate chemistry calculations of the transient start-up environment, and moving/adaptive grids. Turbulence modeling upgrades and their inclusion into PARCH are being performed in a parallel research effort supported by NASA Langley Research Center with progress to date summarized in Refs. 16-20.

Acknowledgements

The authors gratefully acknowledge the Cray computer time provided by the NAS at NASA/Ames for performing many of the calculations described.

References

- ¹Pulliam, T.H., "Euler and Thin-Layer Navier-Stokes Codes: ARC2D, ARC3D," Notes for Computational Fluid Dynamics User's Workshop, The University of Tennessee Space Institute, Tullahoma, TN, UTSI Pub. E02-4005-023-84, March 1984, pp. 14.1-15.85.
- ²Cooper, G.K., "The PARC Code: Theory and Usage," AEDC TR-87-24.
- ³Cooper, G. and Sirbaugh, J., "The PARC Distinction: A Practical Flow Simulator", AIAA Paper 90-2002, June 1990.
- ⁴York, B.J., Sinha, N. and Dash, S.M., "Computational Models for Chemically-Reacting Hypersonic Flow," AIAA Paper 88-0509, Reno, NV, January 1988.
- ⁵Dash, S. M., Sinha, N. and York, B. J. "Matrix Split Approach for Inclusion of SPF Thermochemical Capabilities Into Beam-Warming Based PNS and NS Models", JANNAF 17th Plume Technology Meeting, CPIA Pub. 487, Vol. I, April 1988, pp. 91-110.
- ⁶Sinha, N., York, B.J., and Dash, S.M., "Applications of a Generalized Implicit Navier-Stokes Code, PARCH, to Supersonic and Hypersonic Propulsive Flowfields," AIAA Paper 88-3278, Boston, MA, July 1988.
- ⁷Dash, S.M., "Recent Advances in Parabolized and Full Navier-Stokes Flow Solvers for Analyzing Hypersonic, Chemically-Reacting Flowfield Problems," AIAA Paper 89-1698, Buffalo, NY, June 1989.
- ⁸York, B.J., Sinha, N., Ong, C.C., and Dash, S.M., "PARCH Navier-Stokes Reacting/Multi-Phase Analysis of Generalized Nozzle Flowfields," AIAA Paper 89-1765, Buffalo, NY, June 1989.
- ⁹Sinha, N., York, B.J., Ong, C.C., Stowell, G.M., and Dash, S.M., "3D Navier-Stokes Analysis of High-Speed Propulsive Flowfields Using the PARCH Code," AIAA Paper 89-2796, Monterey, CA, July 1989.
- ¹⁰Dash, S.M., Sinha, N., York, B.J., and Ong, C.C., "3D Navier-Stokes Analysis of Tactical Missile External/Plume Interaction Flowfields," JANNAF 18th Plume Technology Meeting, Monterey, CA, Nov. 1989, CPIA Pub. 530, pp. 21-44.
- ¹¹York, B.J., Sinha, N. and Dash, S.M., "PARCH Navier-Stokes Code Analysis of Rocket and Airbreathing Nozzle/Propulsive Flowfields," JANNAF 18th Plume Technology Meeting, Monterey, CA, Nov. 1989, CPIA Pub. 530, pp. 45-68.
- ¹²Dash, S.M., Sinha, N., and York, B.J., "Computational Models with Advanced Thermochemistry for the Analysis of Missile/Plume Flowfield Interactions," AGARD Symposium on Missile Aerodynamics, Friedrichshafen, Germany, April 23-26, 1990, AGARD CP-493, pp. 31.1-31.30.
- ¹³Dash, S.M., "Advanced Computational Models for Analyzing High-Speed Propulsive Flowfields," 1990 JANNAF Propulsion Meeting, Anaheim, CA, October 1990, CPIA Pub. 550, Vol. I., pp. 247-284.
- ¹⁴Dash, S.M., "Nozzle/Exhaust Plume Code Validation and Improvements in Numerics, Turbulence Modeling, and the Simulation of Nonequilibrium Processes," 19th JANNAF Exhaust Plume Technology Meeting, May 1991, CPIA Pub. 568, pp. 175-202.
- ¹⁵Anderson, L.P., "Transient Pressure Calculations in an Active Canister of the Vertical Launching System," Naval Surface Warfare Center, Technical Report, NSWC TR 82-229, August 1982.
- ¹⁶Dash, S.M., Sinha, N., York, B.J. and Lee, R.A., "Progress in the Development of Advanced Computational Models for the Analysis of Generalized Supersonic Jet Flowfields," AIAA 13th AeroAcoustics Conference, AIAA Paper 90-3915, Tallahassee, FL, October 1990.
- ¹⁷Dash, S.M., "Observations on Practical Turbulence Modeling for High-Speed Jet/Plume Flowfields," AIAA 22nd Fluid and Plasmadynamics Conference, AIAA Paper 91-1789, Honolulu, Hawaii, June 24-26, 1991.
- ¹⁸Dash, S.M., Sinha, N. and York, B.J., "Predictions of Jet Mean-Flow Structure in Support of HSCT Noise-Suppression Concepts," AIAA/SAE/ASME/ASEE 27th Joint Propulsion Conference, AIAA Paper 91-2253, Sacramento, CA, June 24-26, 1991.

¹⁹Dash, S.M., Sinha, N., York, B.J., Kenzakowski, D.C., and Lee, R.A., "Computer Codes for HSCT Exhaust Flowfield Simulation and Observations on Turbulence Modeling," 9th AIAA Applied Aerodynamics Conference, AIAA Paper 91-3297, Baltimore, MD, September 23-26, 1991.

²⁰Dash, S.M., Walker, B.J., and Seiner, J.M., "A Building-Block Approach for Turbulence Model Validation/Upgrade Focussed on High-Speed Jet Flows," to be presented at 1992 JANNAF Propulsion Meeting, February 24-28, 1992.

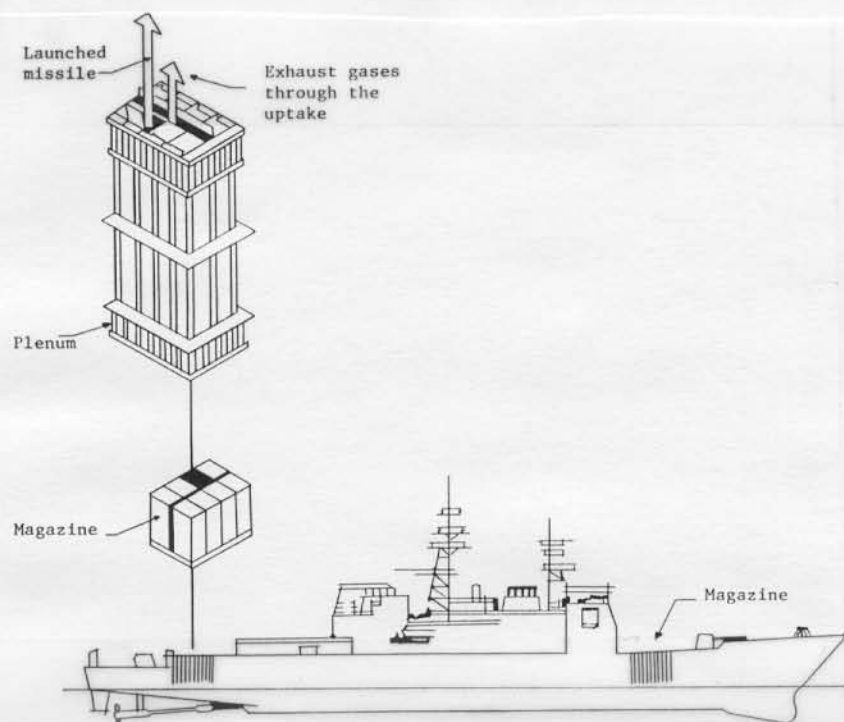


Figure 1. Sketch of a full scale Vertical Launching System (VLS).

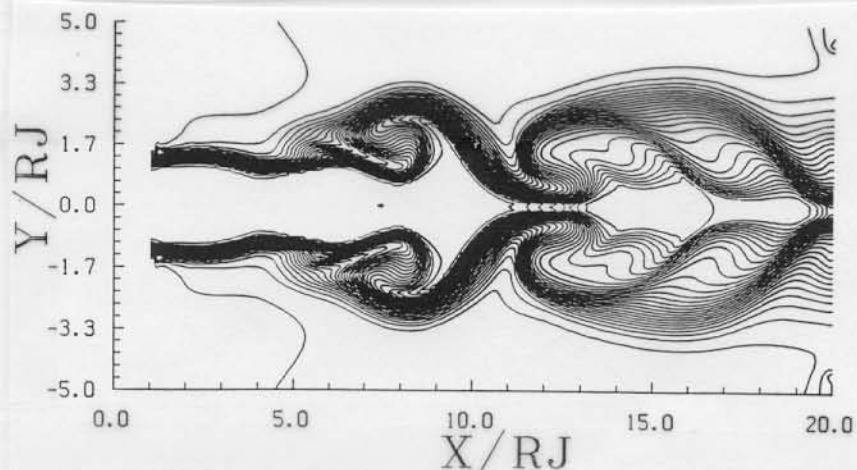


Figure 2. Enthalpy contours from original PARCH2D calculation with loosely-coupled/diagonalized numerics.

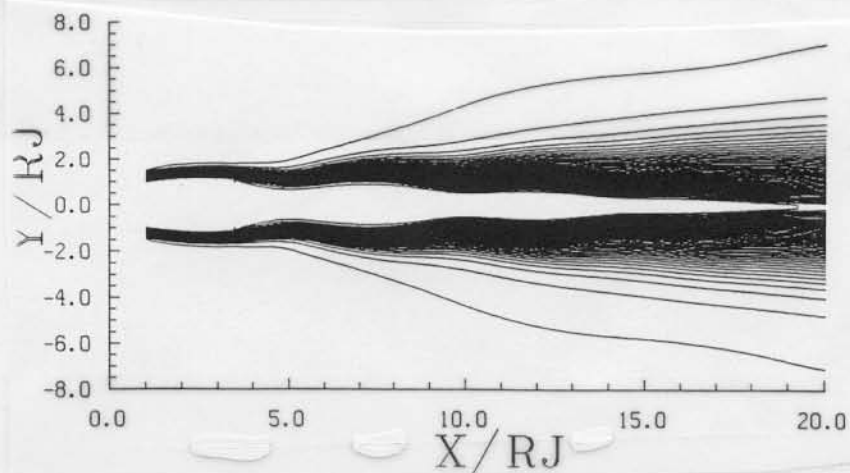


Figure 3. Enthalpy contours from upgraded PARCH/RNP calculation with fully-coupled/block tridiagonal numerics.

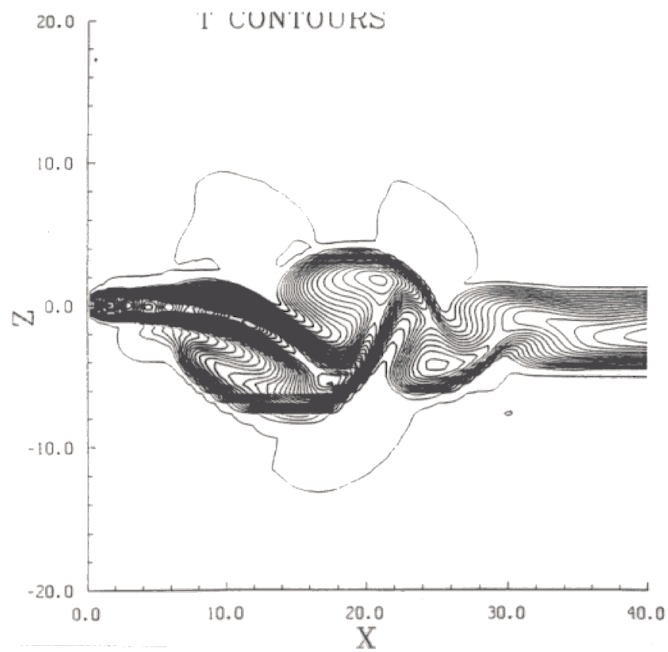


Figure 4. Computed temperature contours in X-Z symmetry plane for 3D jet simulation.

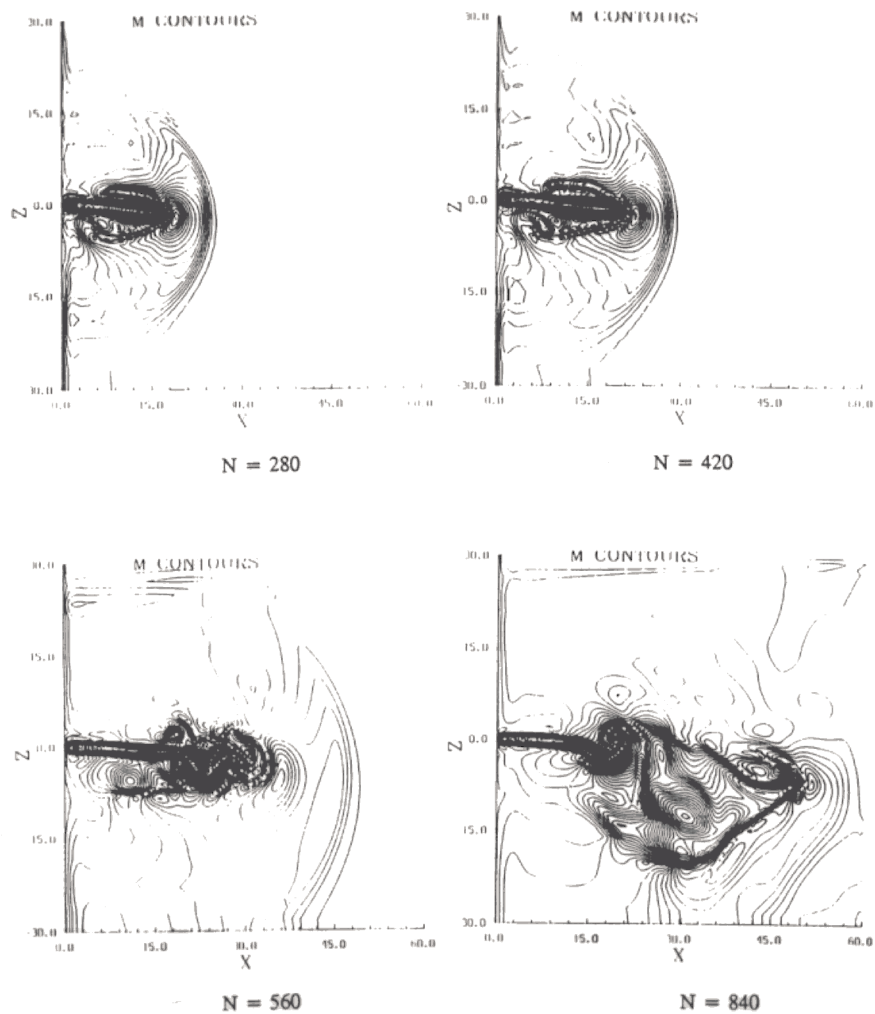


Figure 5. Mach number contours in X-Z symmetry plane at several time intervals for time-accurate jet simulation.

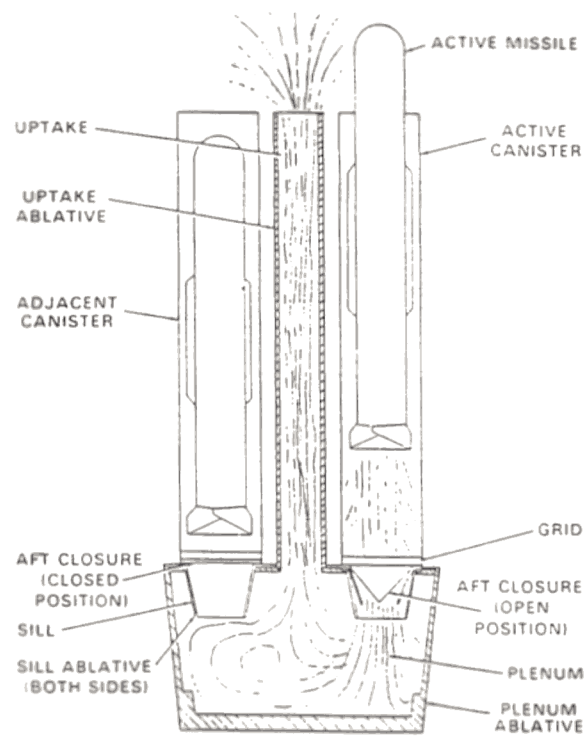


Figure 6. Sketch of the VLS plenum/uptake configuration.

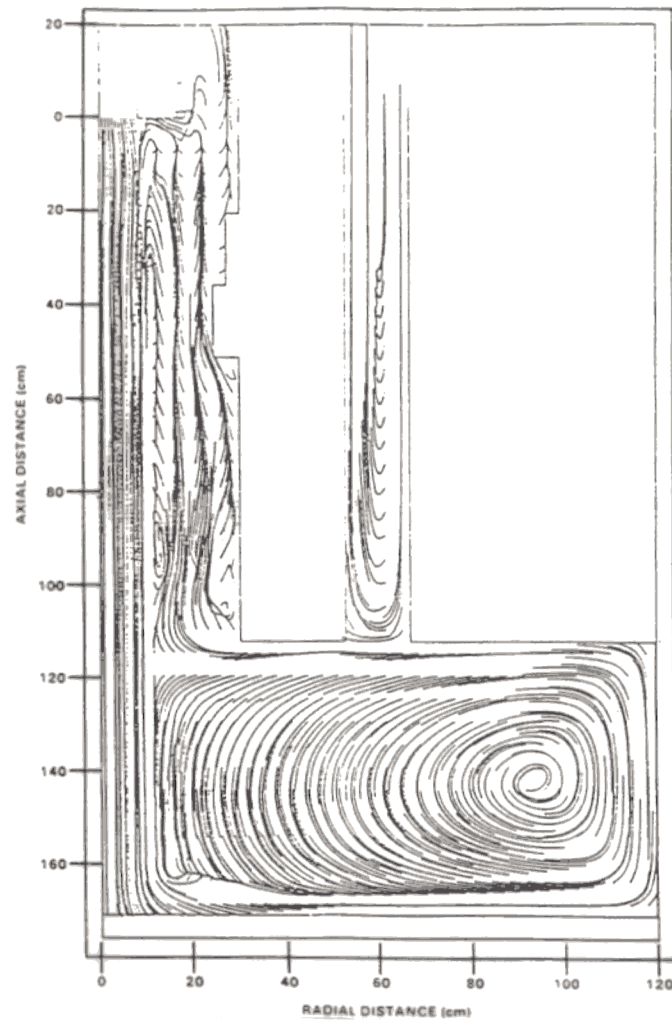


Figure 7. Computed streamlines for 2D VLS plenum/uptake calculation.

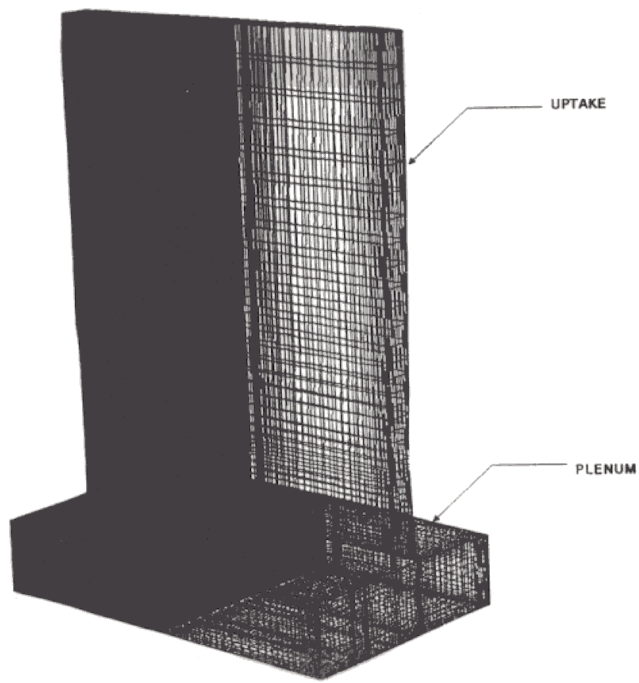


Figure 8. Sketch of simplified VLS geometry for preliminary analysis.

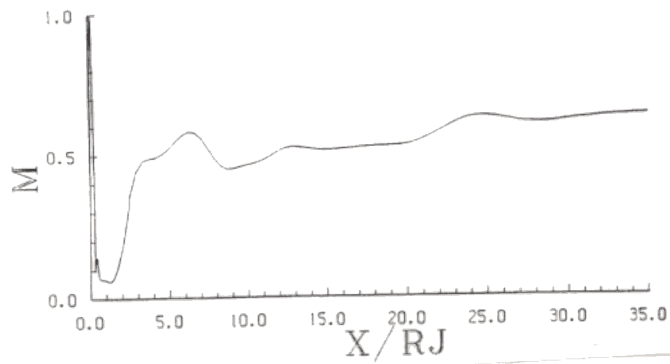


Figure 9. Computed Mach number variation in symmetry plane at center of uptake duct.

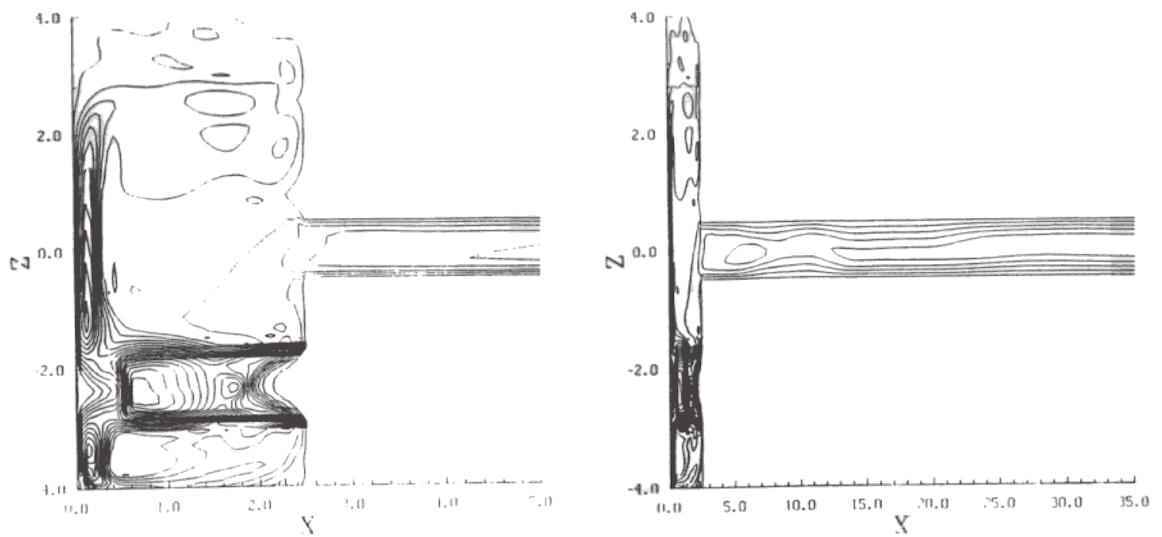


Figure 10. Computed Mach number contours in X-Z symmetry plane for 3D plenum/uptake analysis.

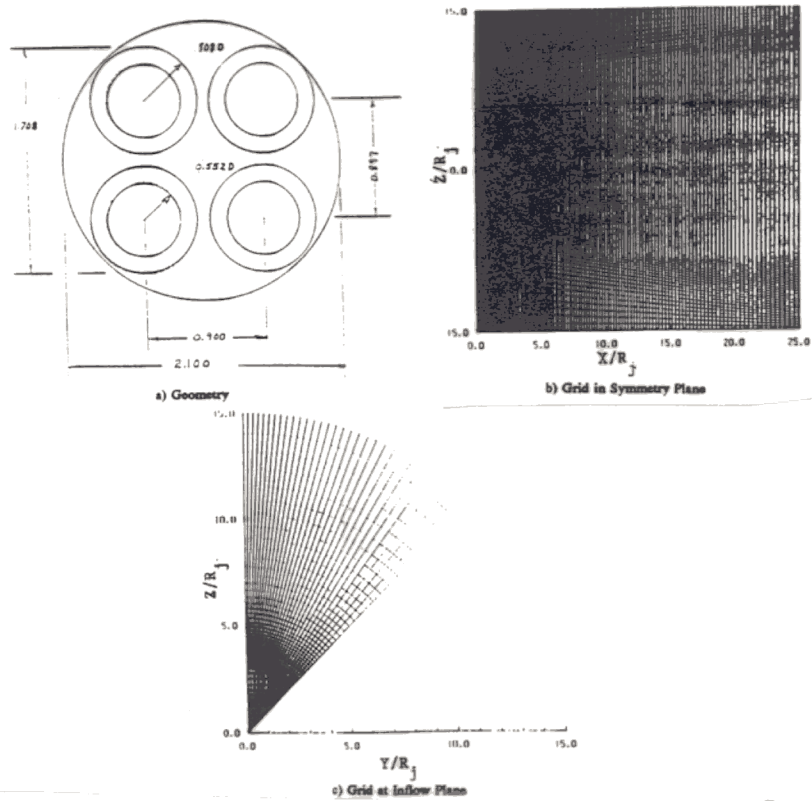


Figure 11. Geometry and computational grid utilized for four nozzle configuration.

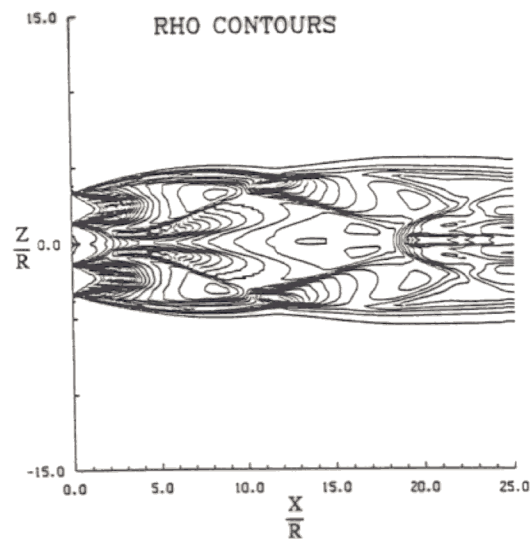


Figure 12. Computed density contours in X-Z symmetry plane for four nozzle simulation.

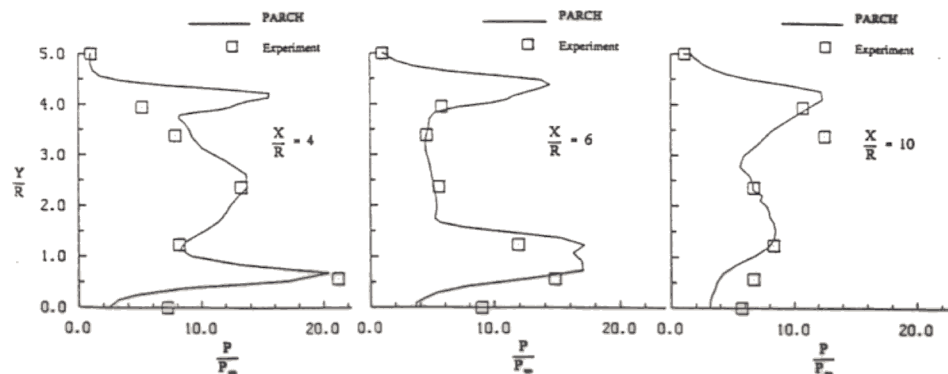


Figure 13. Computed and measured pitot pressure profiles at several axial locations for four nozzle case.