PARC ANALYSIS OF THE NASA/GE 2D NRA MIXER/EJECTOR NOZZLE

J.R. DeBonis
NASA Lewis Research Center
Cleveland, Ohio

S16-07

Interest in developing a new generation supersonic transport has increased in the past several years. Current projections indicate this aircraft would cruise at approximately Mach 2.4, have a range of 5000 nautical miles and carry at least 250 passengers. A large market for such an aircraft will exist in the next century due to a predicted doubling of the demand for long range air transportation by the end of the century and the growing influence of the Pacific Rim nations. Such a proposed aircraft could more than halve the flying time from Los Angeles to Tokyo. However, before a new economically feasible supersonic transport can be built, many key technologies must be developed.

Among these technologies is noise suppression. Propulsion systems for a supersonic transport using current technology would exceed acceptable noise levels. All new aircraft must satisfy FAR 36 Stage III noise regulations. The largest area of concern is the noise generated during takeoff. A concerted effort under NASA's High Speed Research (HSR) program has begun to address the problem of noise suppression. One of the most promising concepts being studied in the area of noise suppression is the mixer/ejector nozzle.

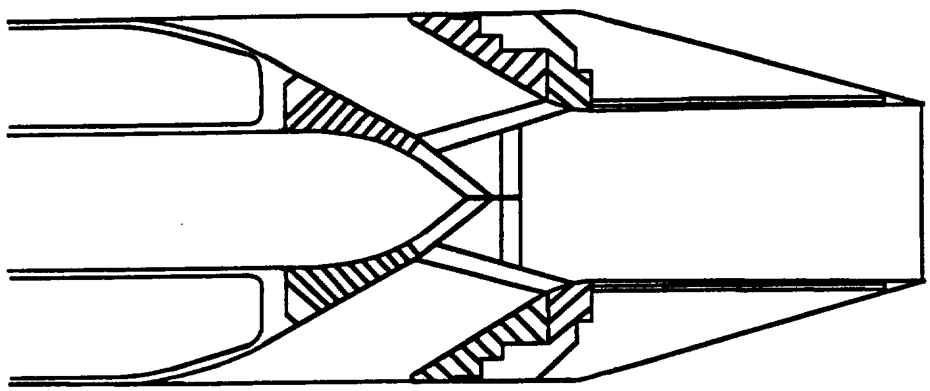
This study analyzes a typical noise suppressing mixer ejector nozzle at take off conditions, using a Full Navier-Stokes (FNS) computational fluid dynamics (CFD) code.

Objectives

- **Analyze the NASA/GE 2DCD mixer/ejector nozzle**
- **Gain a better understanding of mixer/ejector nozzle flow fields**
- **Provide data for improved designs**
- **Evaluate the ability of the PARC code to predict mixer/ejector nozzle flow fields**

The use of CFD can provide valuable information for aerodynamic analysis and design. The objectives of the study are to gain better insight into the nozzle flowfield and provide useful data for improvement of this design and future nozzle designs. Also, by comparing the analytical predictions to experimental data we can evaluate the ability of the CFD code to accurately predict mixer/ejector nozzle flowfields.

NASA/GE 2DCD Mixer/Ejector Nozzle



The General Electric Aircraft Engine Company, under a NASA NRA contract, has designed a two-dimensional (i.e. rectangular) mixer/ejector nozzle for noise suppression. This nozzle is intended to be used in conjunction with a mixed flow turbofan engine.

Mixer/Ejector Nozzles

- **Entrain large amounts of secondary flow**
- **Rapidly mix two flows together to lower jet velocity**
- **Lower jet velocity results in lower noise**
- **Maintain high thrust due to large mass augmentation**

$$F = \dot{m}v$$

Mixer/ejector nozzles entrain large amounts of secondary flow through an array of lobed chutes that are deployed into the primary stream. The low velocity secondary flow is rapidly mixed with the high energy primary flow from the engine to lower the total jet velocity. This lower jet velocity results in lower noise; however high thrust is maintained because of the large amount of flow augmentation.

NASA/GE 2DCD Nozzle

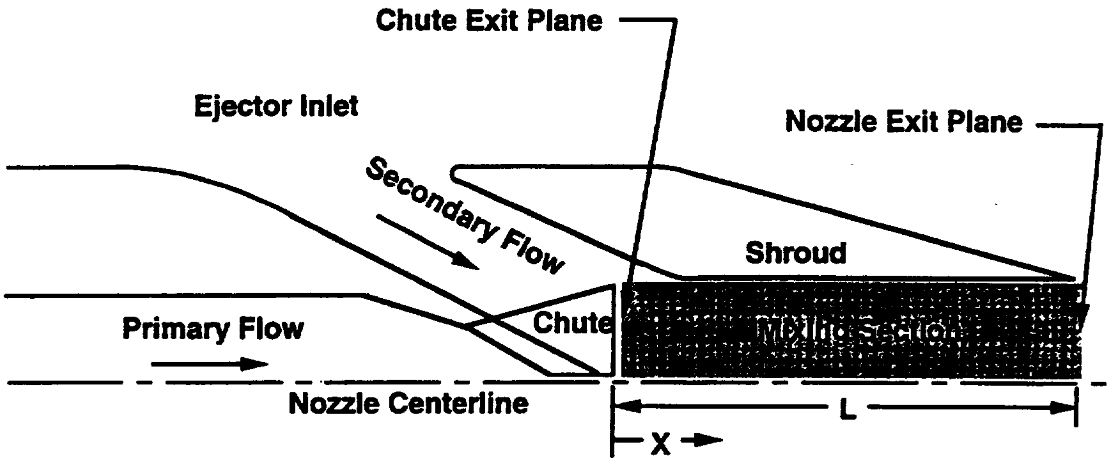
- Rectangular (2D) mixer/ejector
- Short shroud
- SAR = 2.5 (suppressor area ratio)
- Convergent-divergent chutes
- Design secondary flow entrainment of 60 percent
- Test conditions

  $M_\infty = 0.27$ NPR = 4.0

  $T_{op} = 850$ R $T_{os} = 530$ R

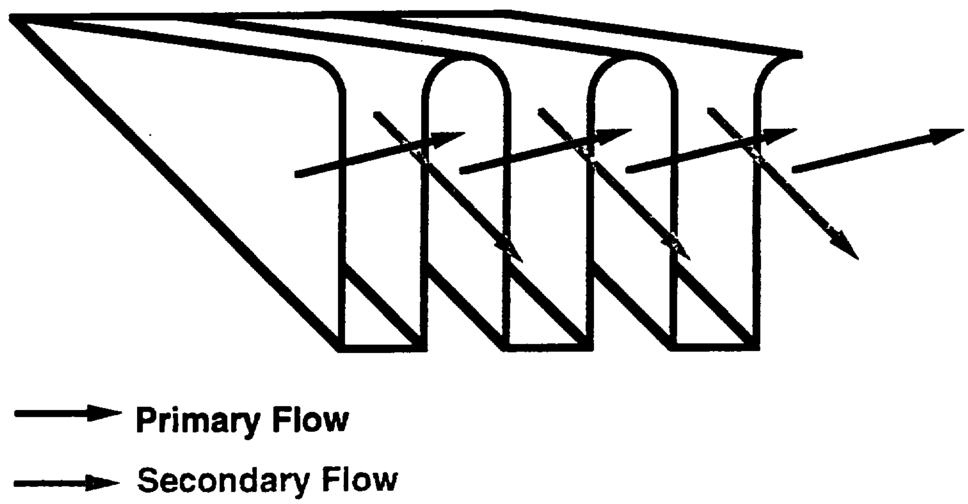
The NASA/GE 2DCD Nozzle is a rectangular mixer/ejector designed for noise suppression. It is designed to entrain approximately 60 percent secondary flow. The nozzle's mixer chutes are a convergent-divergent design. This is intended to eliminate the shock structure in the primary stream. The nozzle studied here is one of several configurations tested in GE's Aerodynamics Research Lab to study its aerodynamic and mixing characteristics. The configuration chosen as the baseline case has a short mixing section and a suppressor area ratio (SAR) of 2.5. The nozzle was studied at the following conditions, NPR = 4.0, $M_\infty = 0.27$, $T_{0p} = 850$ R and $T_{0s} = 530$ R.

Nozzle Schematic



This figure shows the basic flow paths and key elements of the nozzle. The primary flow from the engine passes between the mixer chutes. The secondary flow entrained from the freestream, is drawn into the ejector inlet and through the mixer chutes. At the chute exit plane the two flows meet. A series of streamwise vortices created by the chutes mix the two flows as it passes through the mixing section and exits the nozzle.

Typical Mixer/Ejector Nozzle Chute Geometry



The mixer ejector chutes create the vorticity which mixes the two streams together. These chutes are deployed into the primary stream at takeoff and then retracted when noise suppression is no longer necessary at cruise. The primary flow is directed slightly upward as it moves between the chutes. The secondary flow is drawn down through the chutes and exits them with a downward velocity component. This vertical misalignment of the two flows creates streamwise vorticity at the chute exit plane. This vorticity rolls up into a discrete vortex and stretches as is moves through the mixing section.

Experiment

- Conducted in GE's Aerodynamic Research Lab (ARL)
- Parameters tested
  - ▲ Shroud length
  - ▲ Suppressor area ratio (SAR)
  - ▲ Mixing area ratio (MAR)
- Data includes
  - ▲ Wall static pressures
  - ▲ Kiel probe traverses (P_o, T_o)
  - ▲ LDV measurements

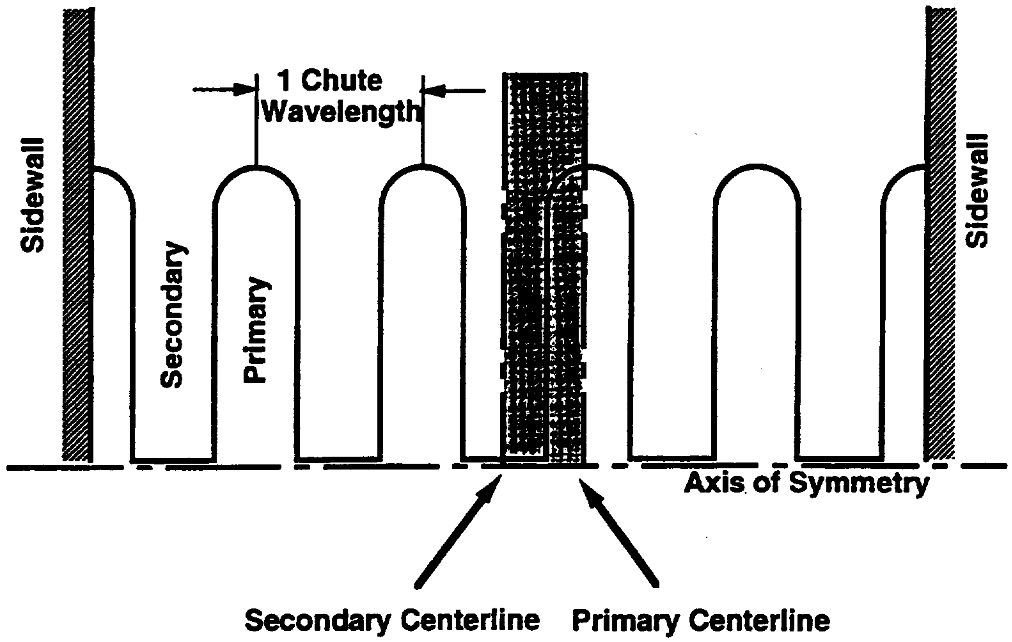
The experimental data was taken at GE's ARL freejet facility. Many nozzle configurations were tested to study the effects of various parameters. These parameters included shroud length, suppressor area ratio, mixing area ratio, and ejector inlet geometry. Mixing area ratio is a measure of the mixing section convergence or divergence and is defined as the ratio of mixing section exit area to mixing section entrance area. Data was taken for a range of nozzle pressure ratio's and freestream mach numbers. The data taken included wall static pressures, Kiel probe traverses of total pressure and temperature and LDV measurements of velocity, flow angle and turbulence intensities.

Grid

- **920,671 grid points**
- **8 Grid blocks**
- **Models 1/2 of a chute wavelength**
- **No sidewall effects**
- **Generated on Iris workstation**
  - **I3G for grid surfaces**
  - **INGRID3D for grid volumes**

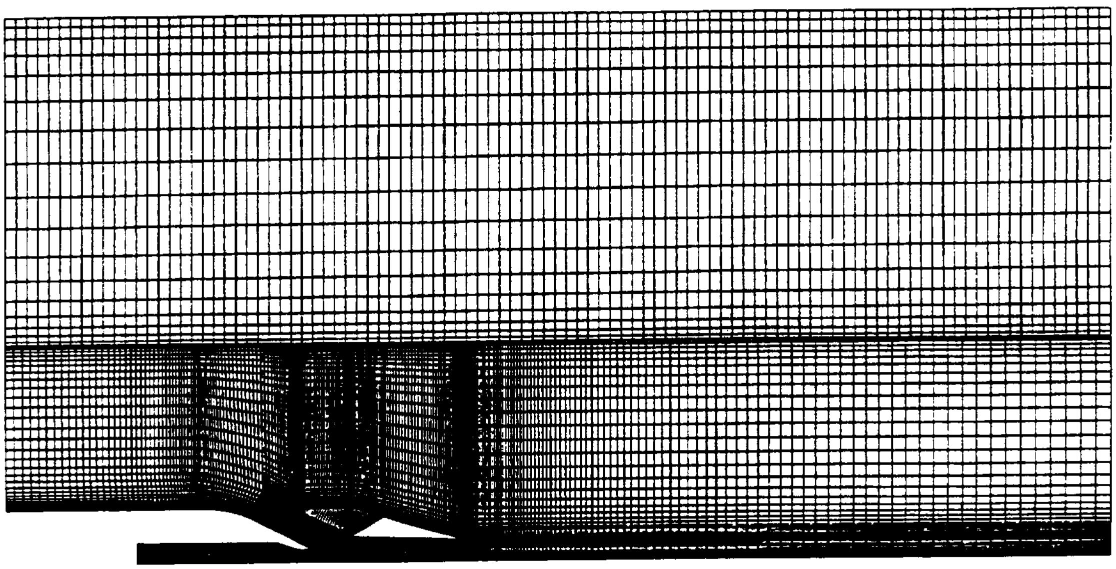
Because of the complexity of the geometry the computational grid is also very complex. The grid consists of 920,671 grid points. This large number of points was necessary to resolve all the internal walls and shear layers present in the flow field. The domain was divided into 8 grid blocks. These blocks divide the geometry such that each individual block is easy to grid. For example, the primary flow path, the chute and the mixing section are all separate grid blocks. Each block is relatively easier to grid than the combined sections. Also, modifications to the grid are made easier, because only the affected grid blocks must be changed. The six surfaces which define a grid block were generated using the I3G interactive grid code. These surfaces were then input into GRIDGEN3D which was used to create the grid volume. The blocks were combined into the completed grid in a post processing step.

Computational Domain



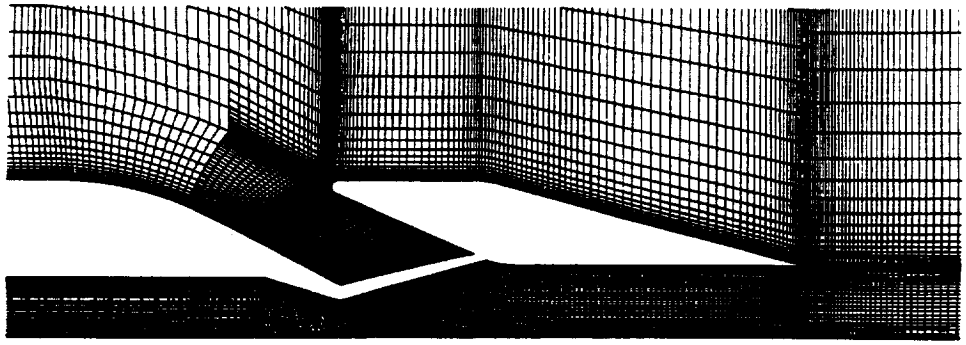
To reduce grid size and computational time, the grid modeled one half of a chute wavelength (defined as the distance from the peak of one mixer lobe to another). Symmetry planes are specified along both the primary and secondary flow centerlines. This is a reasonable approximation for the flow in the center of the nozzle. With this approximation, the effects of the sidewalls are neglected. Also, only the top half of the nozzle is modeled due to the symmetry of the geometry.

Computational Grid

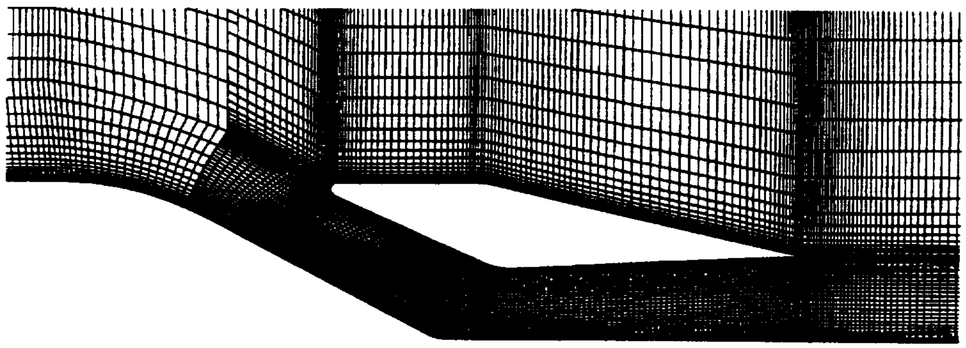


The external flow field as well as the nozzle flow field was modeled. This was done to insure proper calculation of secondary flow entrainment and to study the development of the plume. The external flow was modeled using separate grid blocks. Once the freestream flow has converged, these grid blocks are no longer solved, and the more cpu time can be used on the internal flow field.

Computational Grid
Nozzle Detail



a. Primary Centerline
Ae/Amix = 1.0



b. Secondary Centerline
Ae/Amix = 1.2

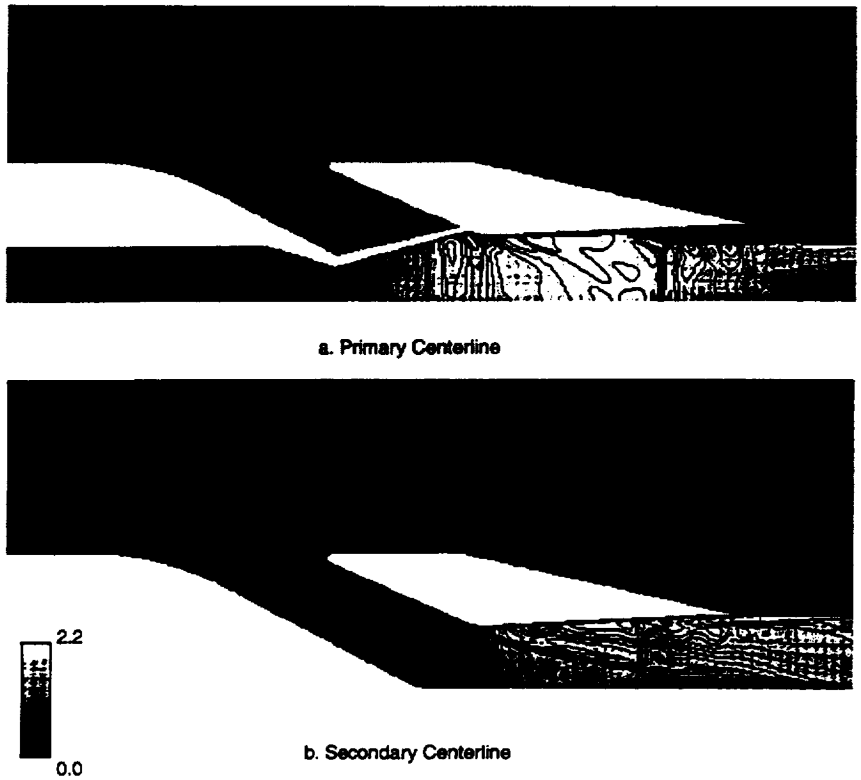
A close-up of the nozzle portion of the grid is shown. Both the primary and secondary flow paths can be seen. Different mixing section area ratios are shown for the primary and secondary flow paths.

The PARC Code

- **Central Differencing**
- **Beam and Warming algorithm**
- **Multiple grid blocks (noncontiguous interfacing)**
- **Generalized boundary conditions**
- **Turbulence models**
  - ▲ **Thomas model (algebraic)**
  - ▲ **K-ε model**

The PARC code is a multipurpose flow solver that was developed at the U.S. Air Force's Arnold Engineering Development Center (AEDC). PARC is central differencing code which solves the Reynolds averaged Navier-Stokes equations using a Beam and Warming algorithm. It has the capability to solve grids made up of multiple grid blocks. The interfaces between blocks do not have to be contiguous. This greatly simplifies grid generation of the multiple blocks. Data is passed between blocks using a trilinear interpolation scheme. Also, the code allows the user to specify any portion of any grid surface as a boundary condition. There are several options available to model turbulence. Both an algebraic model based on the method of P. D. Thomas and a 2 equation K - e model based on a Speziale formulation were used in this analysis.

Mach Number Contours
Ae/Amix = 1.2



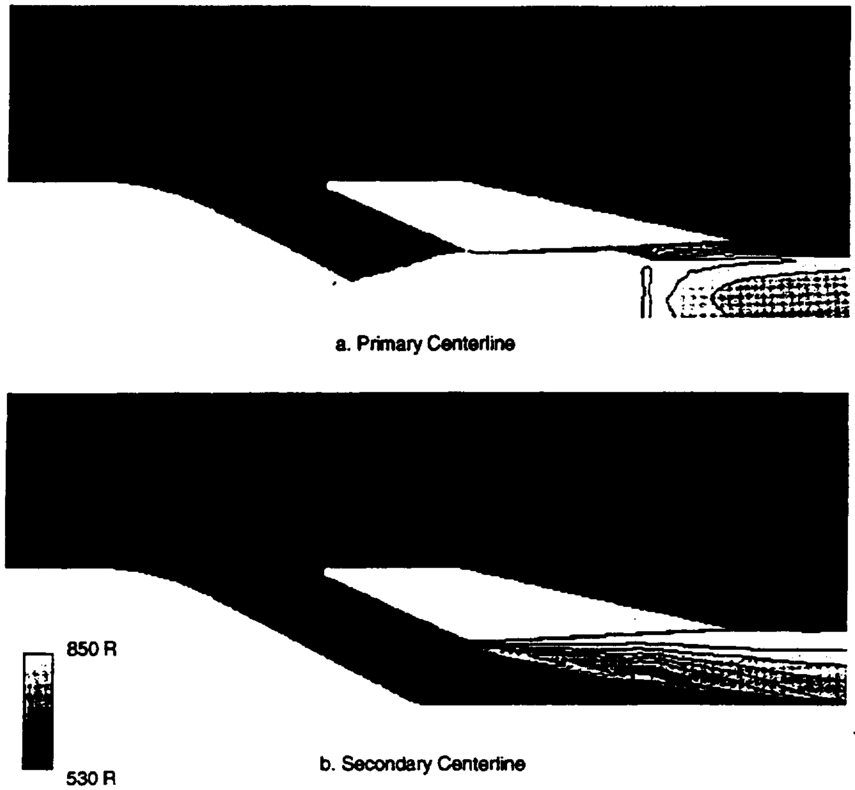
a. Primary Centerline

b. Secondary Centerline

The flow field for the baseline diverging mixing section configuration is presented as a typical flowfield for this nozzle. The primary flow accelerates as it flows between the mixer chutes. The flow chokes just upstream of the chute exit plane and then expands. It undergoes a compression as the flow is turned slightly entering the mixing section. The flow then over expands through the mixing section. The flow shocks near the nozzle exit to reach the ambient pressure. A separation occurs on the shroud wall approximately 60 percent of the way through the mixing section.

On the secondary flow centerline, the flow accelerates through the mixing section and shocks similar to the primary flow centerline. An area of high mach number flow is apparent near the shroud wall and grows in size through the mixing section. No separation is evident on the secondary centerline.

Total Temperature Contours
Ae/Amix = 1.2

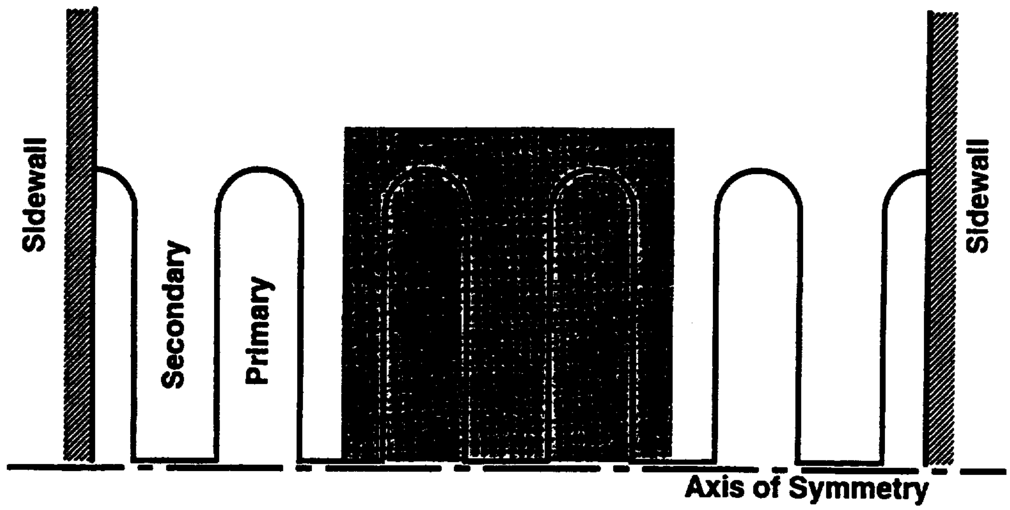


a. Primary Centerline

b. Secondary Centerline

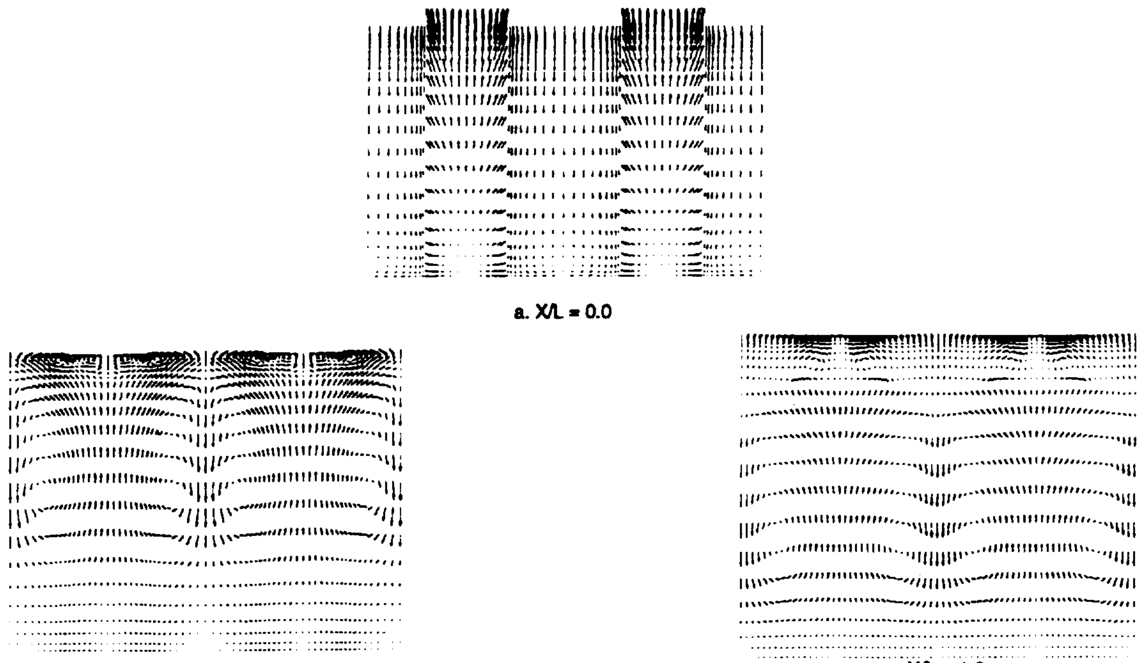
Because the total temperatures of the two streams differ, we can use the total temperature to distinguish the two streams and evaluate the mixing. On the primary centerline the temperature shows very little decay and hence little mixing before the nozzle exit. The separation is evident because the lower temperature ambient air is pulled inside the nozzle by the recirculation. The high temperature flow found on the upper region of the secondary centerline indicates that some primary flow has rolled over into the secondary centerline plane due to the vortical mixing. This explains the existence of the high mach number region show in the previous figure.

Area Plotted



The figure illustrates the area shown for the data plots in the mixing section. The solution has been reflected for clarity to show two complete primary flow passages and one complete secondary passage.

Velocity Vectors
Ae/Amix = 1.2



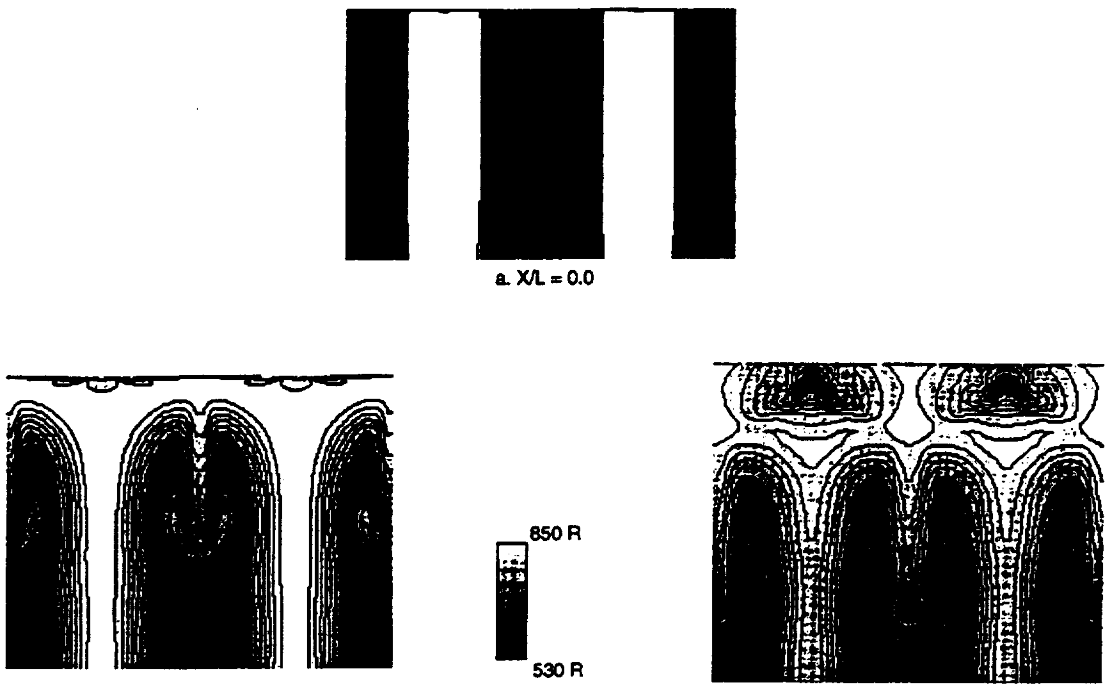
a. X/L = 0.0

b. X/L = 0.5

c. X/L = 1.0

Velocity vectors at three locations in the mixing section show the development of the vortices generated by the mixer chutes. At the chute exit plane (X/L = 0.00), the vertical velocities of the two streams are in opposite directions. This generates a sheet of vorticity along the trailing edge of the chutes. This vorticity rolls up into a discrete vortex in the upper portion of the mixing section. As the flow moves downstream the vortex center moves toward the nozzle centerline and the vortex stretches. At the nozzle exit plane the vortex has stretched to occupy almost the entire exit area.

Total Temperature Contours
Ae/Amix = 1.2

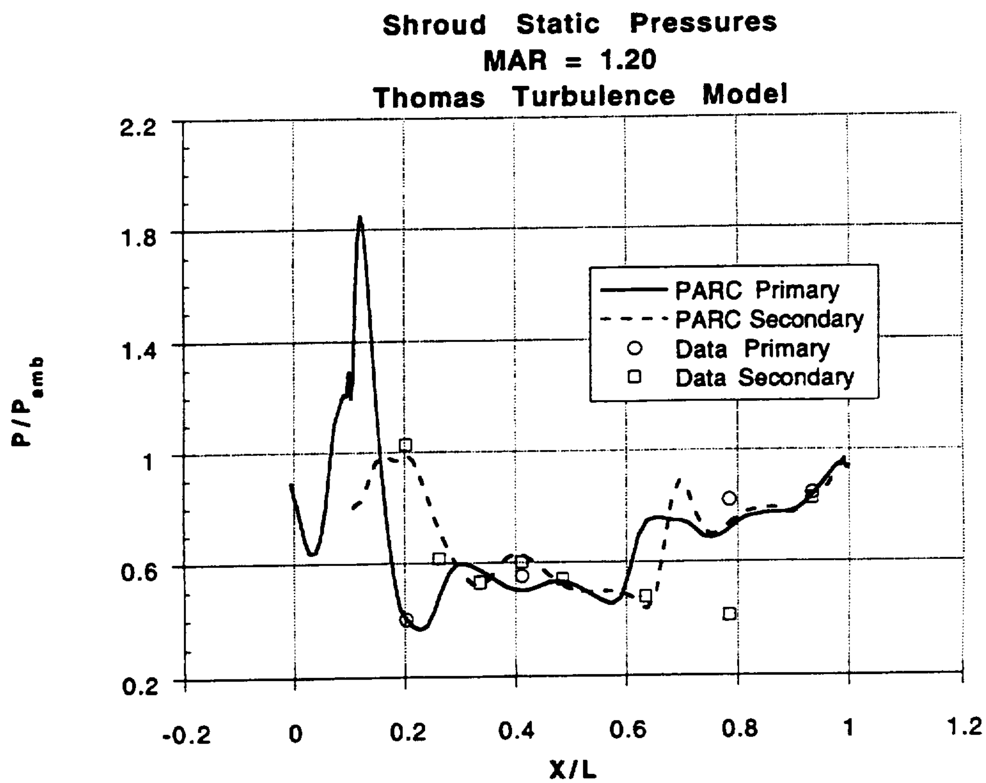


a. X/L = 0.0

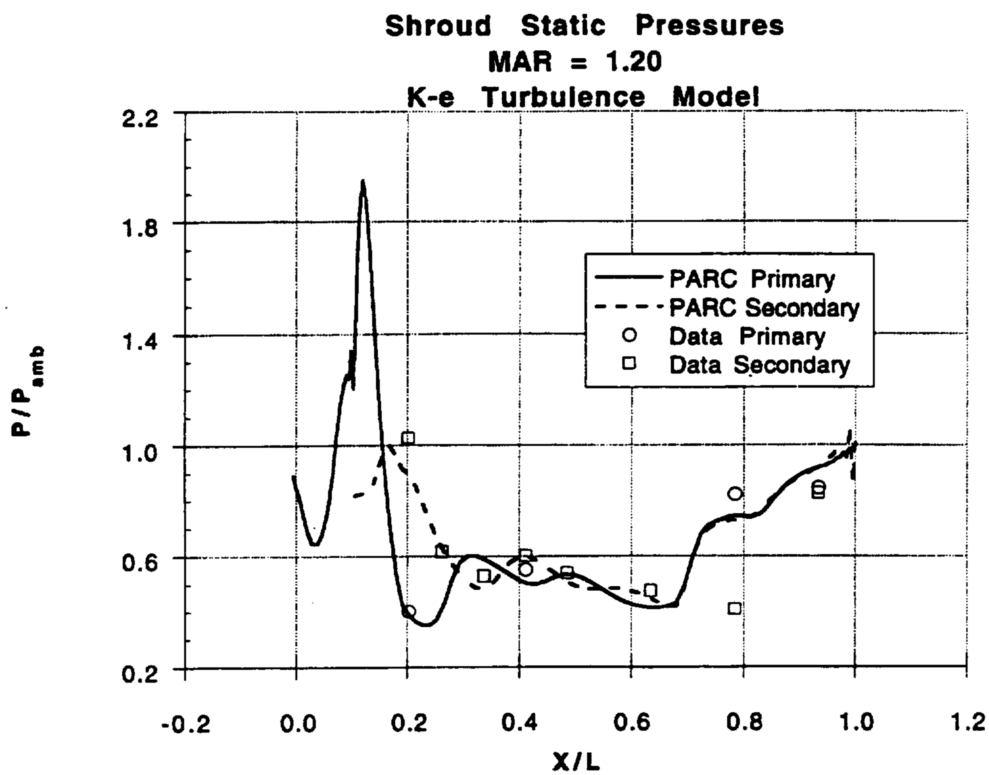
b. X/L = 0.5

c. X/L = 1.0

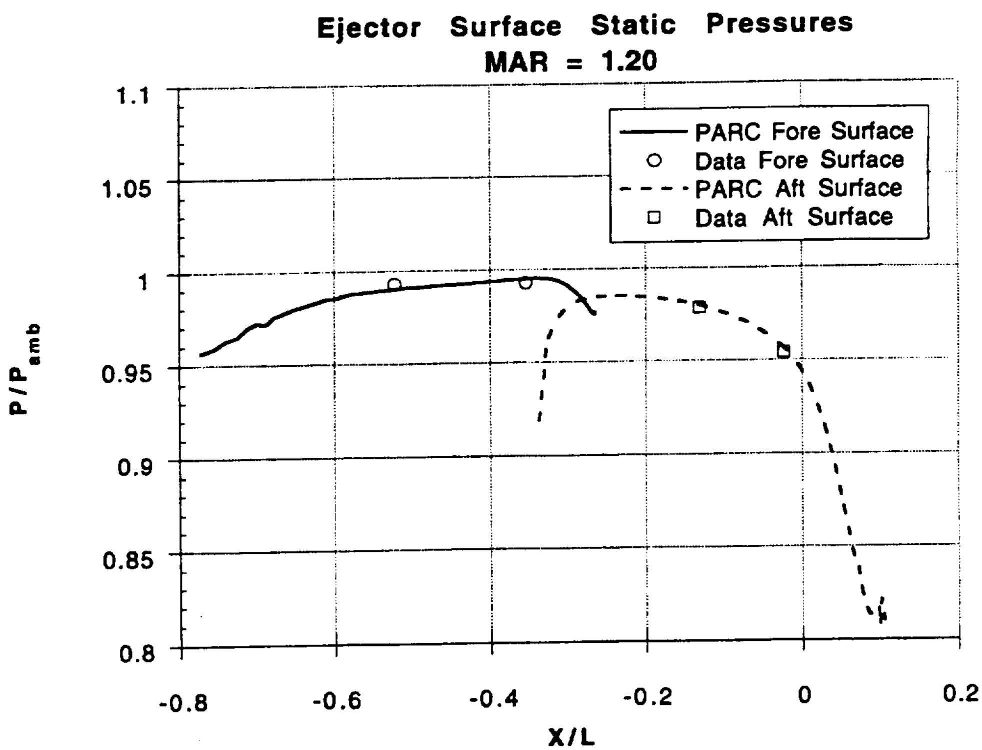
The total temperature contours help visualize the mixing of the streams in the mixing section. The vortex pulls the primary flow over and into the secondary flow plane. As the flow moves through the mixing section, the primary flow continues to migrate into the secondary flow plane and mix with the secondary flow. At the nozzle exit, there are still significant portions of primary and secondary flow that remain unmixed. The separation can be seen near the shroud wall at the nozzle exit. The recirculating flow brings in ambient air which is evident by the lower temperature region near the shroud. This recirculating region occurs only on the primary flow centerline and does not extend across the entire width of the nozzle.



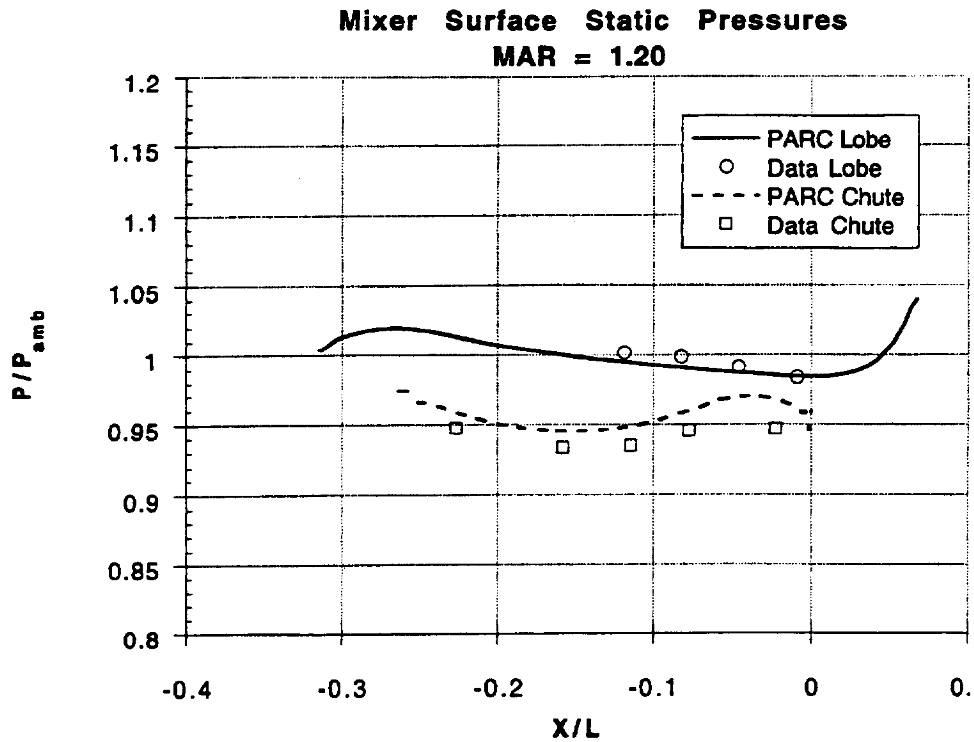
Static pressures on the mixing section shroud walls are presented on both the primary and secondary flow centerlines. The primary flow shocks as it is turned parallel to the shroud wall. Both flows then greatly over expand well below ambient pressure through the mixing section. The flow then shocks and diffuses back to ambient pressure at the exit. Near the region of the separation, the pressures at each location have not yet equalized. This could help explain the localized separation bubble. The predictions agree well with the experimental data. It appears that the PARC code predicts the shock location upstream of the experimental location. This shock has been observed to be unsteady in the experiment and therefore can not be properly resolved using the steady state method of PARC.



The K-e turbulence model predicts a very similar pressure distribution for the first half of the mixing section. The shock is predicted slightly further downstream from the Thomas model data. Also, the static pressures have equalized across the width of the nozzle before the shock. The separation also occurs across the entire nozzle width.

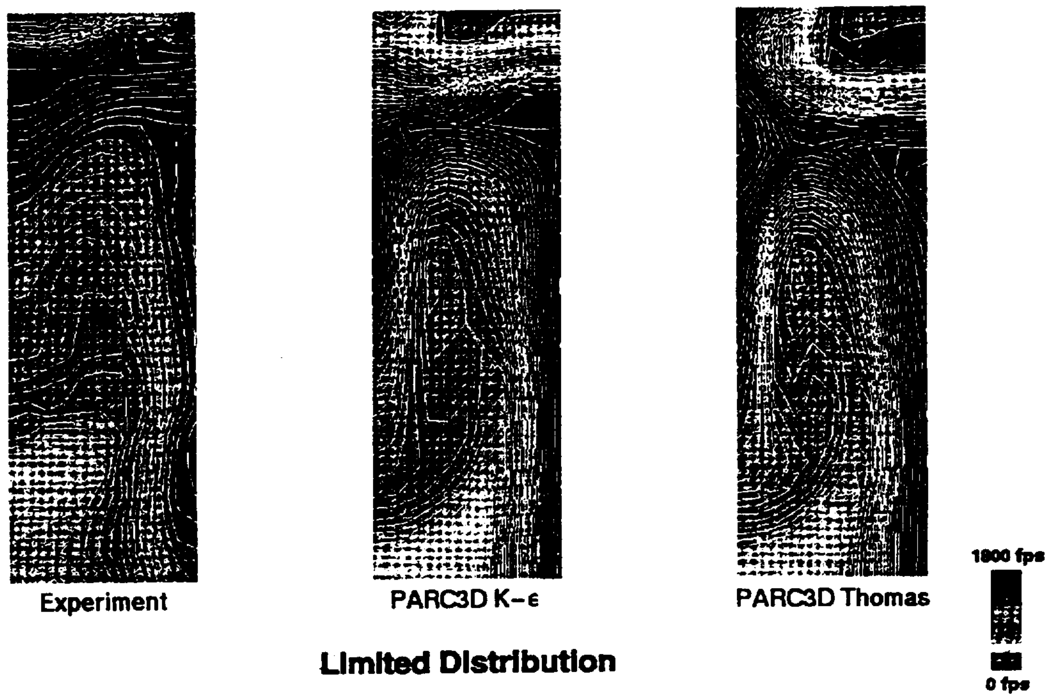


Static pressures on the both the fore and aft ejector surfaces compare very well with experimental data.



Static pressures are shown on both the centerline of the mixer lobe peak, and the centerline of the mixer chute. Agreement is very good on the mixer lobe peak. The prediction is not as good on the chute centerline. However the maximum error is less than 2 percent.

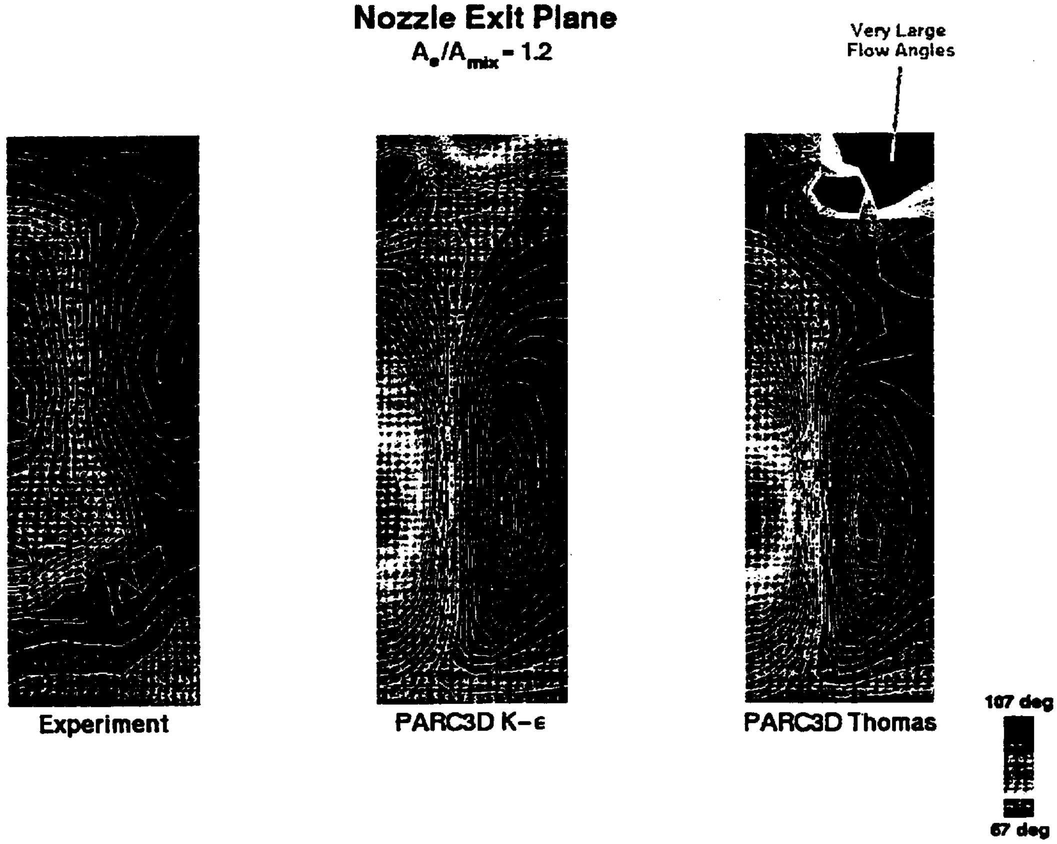
Velocity Contours
Nozzle Exit Plane
$A_e/A_{mix} = 1.2$



Limited Distribution

A 2 component LDV system measured axial and vertical velocities at the nozzle exit plane. The computational results were modified to eliminate the third velocity component. This result was then interpolated onto the experimental grid in order to make a direct comparison. The PARC code has predicted the general trends of the flow field. However, two major differences are observed between computation and experiment. First, the experiment shows a great deal more mixing than predicted by PARC. The K-e solution predicts slightly more mixing than the algebraic model. But, both analytical results greatly under predict mixing. This is most likely a results of the turbulence models used. Also, in the experiment upstream flow disturbances not modeled in the analysis may have been present which could have aided in mixing. The second major difference between analysis and measurement is in the separated region on the shroud wall indicated by a very low velocity region in the upper portion of the contours. Both turbulence models show that the separated region still exists at the exit plane. The experiment seems to infer that the flow has reattached by the exit plane. The prediction of reattachment downstream of the actual location is typical of the PARC code. The K-e model predicts a thinner separated region than the Thomas model. The Thomas model solution shows that the separation does not span the entire width of the nozzle and is somewhat unrealistic.

Flow Angle Contours
Nozzle Exit Plane
$A_e/A_{mix} = 1.2$



Experiment PARC3D K-ε PARC3D Thomas

The flow angles presented here are defined as the angle the 2D velocity vector makes with the vertical plane; 0 degrees down, 90 degrees axial. The flow angles also indicate that the PARC code has predicted less mixing and late separation reattachment. The vortex appears as two parallel elliptical areas with opposite flow direction. The predicted size of these regions agrees well with the data. The experimental position of the vortex is closer to the shroud wall than predicted by PARC. This is probably due to the separation region still remaining in the analysis forcing the vortex away from the wall. The Thomas model predicted very large flow angles in the recirculating region. These large angles were neglected in order to make a clear comparison.

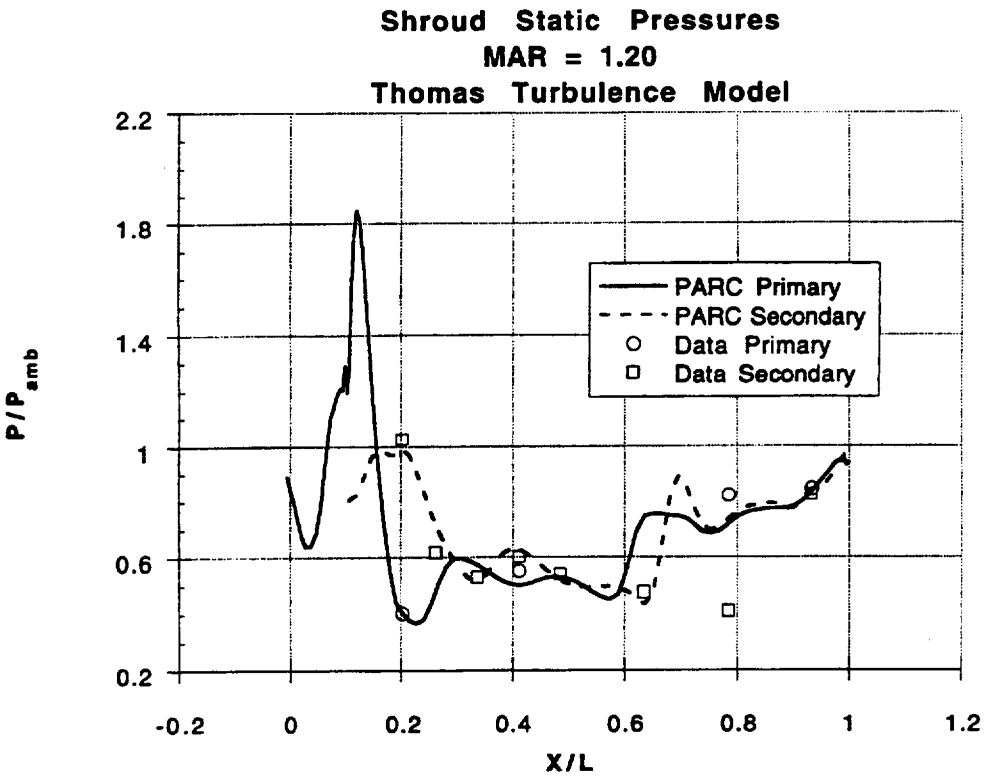
MAR Study

- Current Designs operate over expanded
- Determine the optimum Mixing Area Ratio (MAR) for nozzle performance
- Modified existing grid
- Used previous solution as initial solution
- Four configurations studied

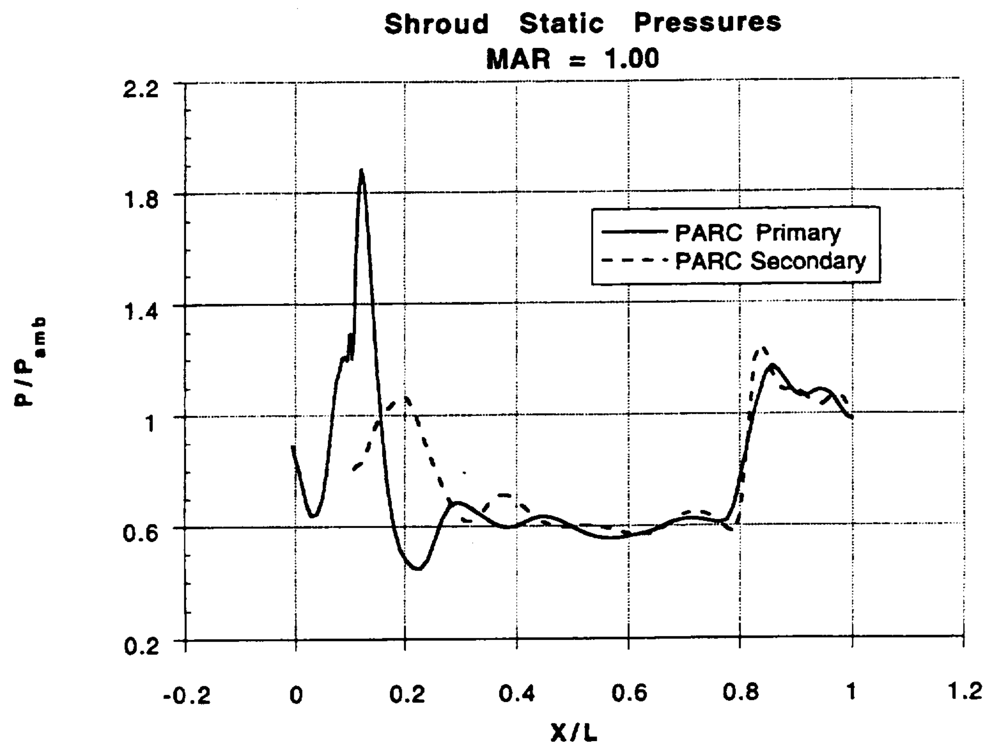
$$\left.\begin{array}{l}\blacktriangle\ \text{MAR} = 0.90 \\ \blacktriangle\ \text{MAR} = 0.95\end{array}\right\}\ \text{convergent}$$

▲ MAR = 1.00 - constant
▲ MAR = 1.20 - divergent

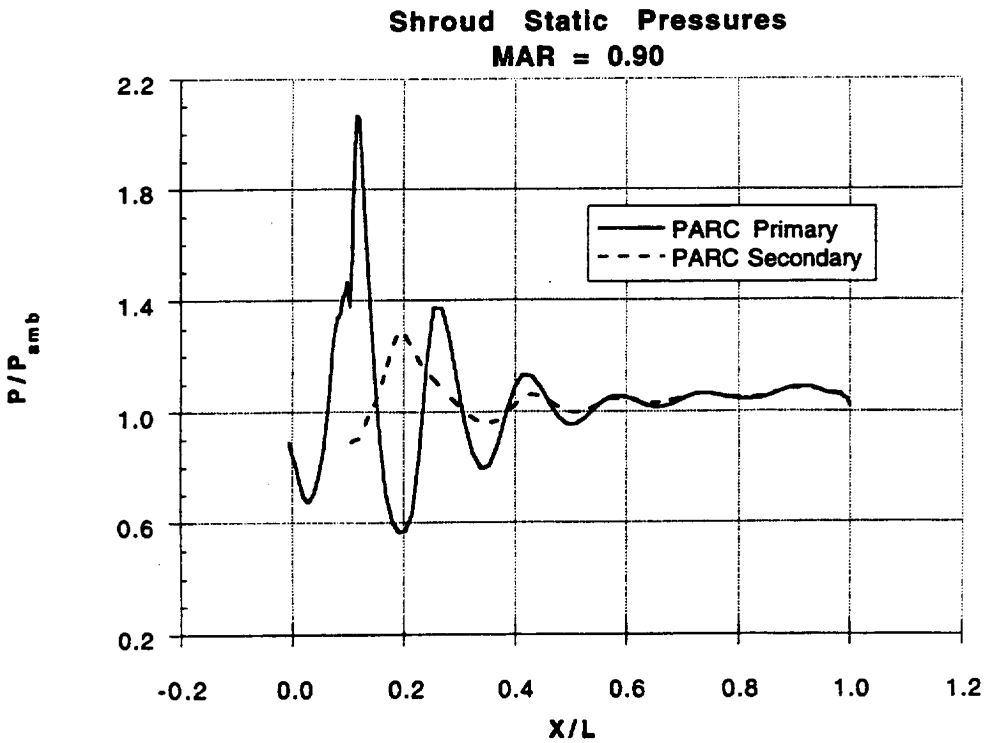
The flow in the mixing sections of the nozzle configurations tested in ARL was over expanded and thus had poor thrust performance. In order obtain maximum thrust performance for this nozzle, a study was done to determine the optimum mixing area ratio (MAR). Because the grid was generated in multiple blocks, only the affected blocks had to be modified. This greatly simplified the grid generation process. A completed solution was used as the initial conditions for the new case. This decreased the number of iterations necessary for convergence. Four cases were run to find the optimal MAR value. They were; 1.20, 1.00, 0.90, and finally 0.95.



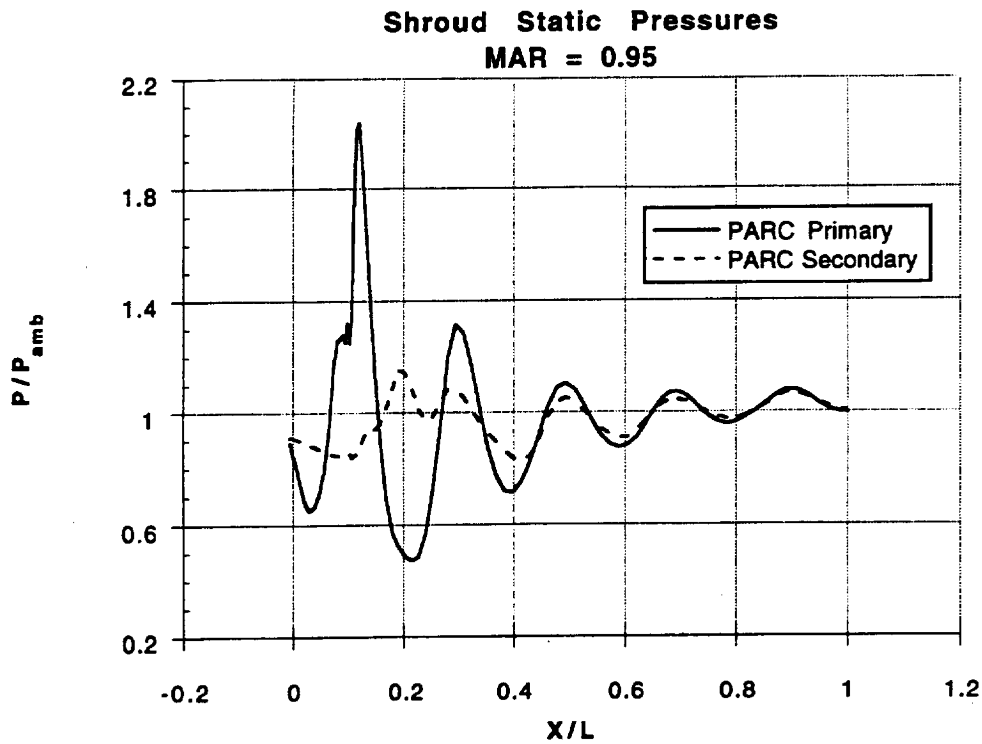
Static pressures on the shroud surface are presented to show the effect of the mixing area ratio on the flow expansion. For the baseline case, MAR = 1.20 the flow greatly over expands to under 50 percent of ambient pressure. To match ambient pressure at the nozzle exit the flow shocks. The large divergence of the shroud also causes the flow to separate from the shroud wall. The shock wave is not clearly defined by the wall pressures due to the large separation.



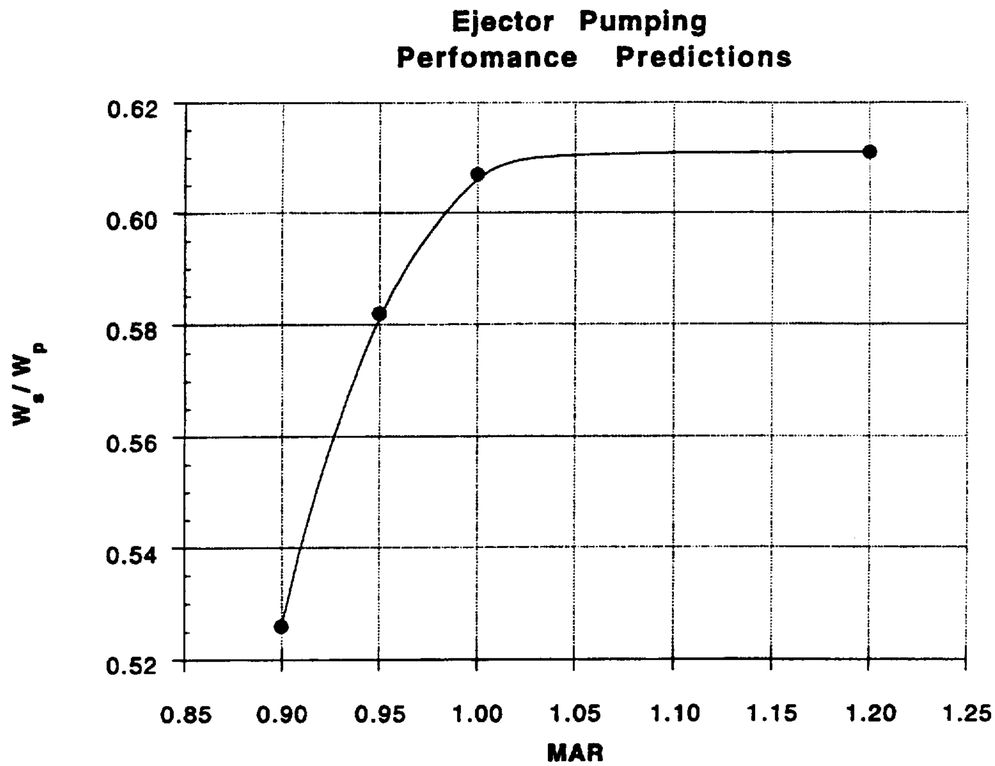
The constant area mixing section also shows an over expansion. The resulting shock can be clearly seen because no separation was evident.



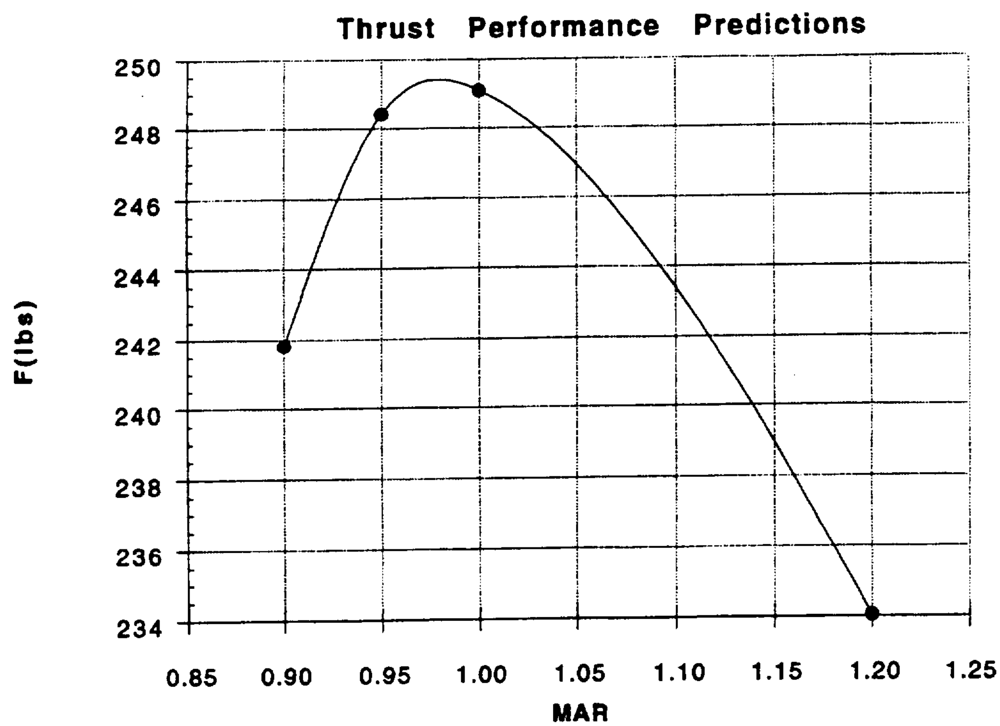
For the first converging case analyzed the flow appears to be slightly over expanded. Mass flow augmentation was reduced significantly.



The final case run was MAR = 0.95. The pressure distribution shows that the flow contains a series of oblique shocks in the mixing section. At the nozzle exit the flow is near ambient pressure.



The effect of the mixing area ratio on ejector pumping can be seen in this figure. For a MAR greater than 1, the secondary flow is choked and the exit area of the nozzle has no influence on amount of flow entrained. The amount of flow entrainment meets the goal value of 0.60 for mixing area ratios greater than 0.98. For the converging cases, the secondary flow is not choked and the reduction in nozzle exit area reduces the amount of secondary flow which is entrained.



The thrust vs. mixing area ratio curve show a definite peak near MAR = 0.97. As MAR is increased beyond this point thrust is lost due to over expansion and eventually separation. For a MAR less than 0.97 thrust is lost due to a reduced amount of secondary flow and under expansion. The thrust values presented are pure thrust and do not take into account any drag penalties.

Conclusions

- **PARC code accurately predicts major flow features**
- **K-e turbulence model gives some improvement in separated regions**
- **PARC under predicts the extent of mixing**
- **Optimum nozzle performance at MAR = 0.97**

Mixer/ejector nozzles have the potential to lower jet noise without significant thrust loss. A full Navier-Stokes analysis of the NASA/GE 2DCD mixer/ejector nozzle was performed. The PARC code predicts with good accuracy the basic flow field of the nozzle. Pressure distributions compare very well with experimental data. However, the PARC code under predicts the extent of the primary and secondary flow mixing. The two equation K-e turbulence model and the algebraic Thomas model produce very similar results. But the K-e model does produce more realistic results in the separated region. A study to determine the mixing area ratio for best thrust performance concluded that this MAR should equal 0.97.