Experimental Investigation and Computation of a Supersonic Ejector Nozzle with Clamshells

Deepak Thirumurthy\textsuperscript{1}

\textit{Energy Division, Rolls-Royce North America, Mount Vernon, OH 43050}

Jesse T. Jones\textsuperscript{2}

\textit{Missile and Aviation Systems Division, Dynetics, Huntsville, AL 35806}

Gregory A. Blaisdell\textsuperscript{3}, Anastasios S. Lyrintzis\textsuperscript{4} and John P. Sullivan\textsuperscript{5}

\textit{School of Aeronautics and Astronautics, Purdue University, West Lafayette, IN 47906}

A new supersonic ejector nozzle with clamshell doors is proposed as a noise suppression jet engine exhaust system. A design table driven, parametric nozzle geometry was designed. The experimental and numerical studies of its flow field were carried out. Cases with and without clamshells were considered and their mean velocity flow fields were compared. The experimental investigation involved the testing of the nozzle in a wind tunnel and the measurements were taken using a seven-hole probe, mounted on an automated 2-axis traverse instrument. Various flow visualization techniques such as the fluorescent oil flow, surface sediment traces, smoke wand and surface tuft were also implemented to capture the flow physics on the inside of the clamshells. Numerical simulations were performed using the commercially available finite volume-based computational fluid dynamic code FLUENT. The computational results are in good agreement with the experimental measurements at different axial locations downstream of the nozzle throat. A zone of flow separation and recirculation is captured at the inner surface of the clamshells in both the experimental investigation and numerical computation.

Nomenclature

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAD</td>
<td>Computer-aided-design</td>
</tr>
<tr>
<td>C-D</td>
<td>Convergent-divergent</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Lewy number</td>
</tr>
<tr>
<td>CNC</td>
<td>Computed numerically controlled machine</td>
</tr>
<tr>
<td>FA</td>
<td>Fundamental aeronautics program</td>
</tr>
<tr>
<td>FAR</td>
<td>Federal aviation regulation</td>
</tr>
<tr>
<td>HST</td>
<td>High speed transport program</td>
</tr>
<tr>
<td>IGES</td>
<td>Initial graphics exchange specification format</td>
</tr>
<tr>
<td>NPR</td>
<td>Nozzle pressure ratio</td>
</tr>
<tr>
<td>PAG</td>
<td>Polyalkylene glycol</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-averaged Navier-Stokes</td>
</tr>
<tr>
<td>RMS</td>
<td>Root mean square</td>
</tr>
<tr>
<td>SCAR</td>
<td>Supersonic cruise aircraft research program</td>
</tr>
<tr>
<td>SST</td>
<td>Supersonic transport program</td>
</tr>
<tr>
<td>SST</td>
<td>Shear stress transport turbulence model</td>
</tr>
</tbody>
</table>

\textsuperscript{1}\textit{Mechanical Engineer, Member AIAA.}
\textsuperscript{2}\textit{Missile Systems Engineer, Member AIAA.}
\textsuperscript{3}\textit{Associate Professor, Associate Fellow AIAA.}
\textsuperscript{4}\textit{Professor, Associate Fellow AIAA.}
\textsuperscript{5}\textit{Professor, Associate Fellow AIAA.}
I. Introduction

The quest to fly faster than the speed of sound has revolutionized the aircraft design. Designs of aircraft have changed from subsonic swept wings to blended wing bodies for better aerodynamic efficiency and reduced drag. Similar trends can be seen in air-breathing propulsion system design. The design requirements for a viable supersonic transport pose conflicting requirements on the design and configuration of the propulsion system. Most of the high speed propulsion system components vary significantly in their design and performance when compared to their subsonic counterparts, and exhaust nozzles are no exception. The performance of the exhaust nozzle is critical to the overall system performance as it provides the required thrust efficiently during different phases of flight such as subsonic takeoff, transonic acceleration, supersonic cruise and subsonic approach. In addition to performance issues, noise associated with the high speed exhaust jet is a concern in modern supersonic cruise nozzles. The aircraft engine has to comply with the Federal aviation regulation (FAR) Stage III noise regulations during all phases of flight. This poses additional requirements in the design and performance of jet engine exhaust systems.

During the past 50 years, several high speed propulsion systems have been designed to address these challenges. Research and development programs such as supersonic transport (SST), supersonic cruise research (SCAR), high speed transport (HST), and fundamental aeronautics (FA) were initiatives to address problems related to supersonic civil transport. During these programs, special emphasis was given to the jet engine exhaust system design. Many noise suppression approaches were studied and combinations of technologies were formulated into acoustically effective, “optimum” configurations based on quantitative analyses. At the supersonic cruise point, the lift to drag (L/D) ratio of the aircraft is low and the specific fuel consumption is high relative to subsonic jetliners. The aircraft payload weight thus becomes highly sensitive to nozzle efficiency. For example, for the Concorde, at the cruise speed of Mach 2.2, a 1% decrease in nozzle performance was estimated to be equivalent to an 8% loss in payload weight.

Engine manufacturer Rolls-Royce and business class aircraft manufacturer Gulfstream Aerospace Corporation are collaborating on the development of technologies for a supersonic jet. A similar ejector nozzle configuration was used in the Olympus-593 engine which powered the Concorde aircraft. In the past, Rolls-Royce conducted studies to understand the noise sources associated with supersonic transport and to improve the noise suppression capability of ejector nozzles without compromising their cruise performance.

II. Ejector Nozzle Exhaust System

In the past, ejector nozzles were introduced primarily to minimize jet noise while making use of their thrust augmentation capability. Ejector nozzles act as blow-in doors, introducing tertiary air mass flow from the atmosphere. This flow is mixed with the primary flow (hot flow) and secondary flow (cold flow) to reduce the exhaust jet velocity without compromising the net thrust. The resulting decrease in the jet exhaust velocity ensures noise reduction.

The concept of ejectors (or injectors) is not new and dates back to the early 19th century when Henri Giffard demonstrated the potential of ejectors and filed a patent. An ejector, in the context of nozzle aero-
dynamics, is a means of pumping a stationary or low speed atmospheric flow into the high speed nozzle flow, primarily due to the momentum transfer between the mixing streams. Therefore, the ejector performance and its noise characteristics are dependent on the turbulent mixing which takes place in a free shear layer originating from the end of the separation between the primary and secondary flows.\(^6\)

As studied by Der\(^6\) in detail and shown in Figure 1, the ejector performance in general, can be deduced from the behavior of the associated shear layer and its relation to the ejector shroud walls. For instance, if the mixing duct length \((L)\) is too small, the shear layer will not attach to the shroud wall, leaving the secondary flow open to a stronger influence from external conditions and susceptible to separation and recirculation. On the other hand, if \(L\) is long and extends beyond the attachment point of the shear layer with the shroud walls, frictional losses will result.

Some of the earlier applications of ejectors in jet propulsion were limited to the cooling of afterburning exhaust systems. Huddleston et al.\(^7\) studied the pumping of air for the cooling of turbojet exhaust systems. As stated by Kochendorfer and Rousso,\(^8\) much of the ejector performance data available as late as 1951 was inapplicable to jet engine ejector design. At that point, most of the existing work still focused on industrial applications with significantly different geometries and flow ratios. Most of the tested designs were either axisymmetric or had basic rectangular flow paths, so as to be well defined by the various geometry parameters, easing comparison with each other. Hence a more comprehensive study of the performance of a 3-D asymmetric ejector nozzle has always been a requirement.

The potential of ejectors in thrust augmentation and jet noise reduction (as opposed to cooling) was identified in 1960s during various SST programs. From 1960 to 1985, several supersonic transport programs such as SST, SCAR and HST were started and several new ejector-based nozzle concepts were realized. In general, two types of ejector nozzles were examined during these programs. The first, termed an auxiliary door ejector, involved small doors which enabled the external atmospheric air to enter into the nozzle flow through ducts. The second, called a variable flap ejector, functioned by controlling the size of the secondary passage and mixing shroud angle through a variable flap. A comprehensive summary of various nozzles designed during these supersonic transport programs and their experimental results are given by Stitt.\(^3\)

Almost all of the ejector nozzle concepts studied before the 1970s were limited to the research level and never materialized in practical supersonic air-transport applications. The only exception to this is the exhaust nozzle system of the Rolls-Royce Olympus-593 engine, which powered the world’s first supersonic airliner, the Concorde.\(^9\) This design serves as the key motivation to the present study of an ejector nozzle with clamshells. In addition to thrust augmentation and noise suppression, the clamshell-based ejector nozzle also serves other key nozzle operational functions such as independent variation of exit area and reverse thrust operation. During take-off at subsonic speed, the clamshell angles are scheduled as per the exit area requirement for optimum performance. During supersonic cruise, the clamshells are completely closed, thereby representing a conventional convergent-divergent (C-D) nozzle and giving high efficiency. After landing, the clamshells can be deployed for the application of reverse thrust. These three operations are shown in Figure 2.

III. Experimental Investigation\(^{10}\)

The aim of the project was to develop a unique test model that captured some of the fundamental aerodynamic features of an existing design. This also facilitated in exploring proposed modifications, enabling fabrication and function of an effective test model. The result of this effort was a test nozzle of 0.123 scale operated at approximately \(M_{\text{throat}} = 0.25\) and \(Re_D = 760,000\). This corresponds to a nozzle pressure ratio (NPR) of the order of 1.065.

A. CAD Geometry and Experimental Setup

To fabricate the test nozzle geometry utilizing the automated computer numerical controlled (CNC) machining techniques required by such complex geometry, it was necessary to start with a high fidelity computer-aided-design (CAD) definition. Such a CAD model also enables collaboration with other members of the research team pursuing computational fluid dynamic (CFD) analysis. A design table driven, parametric CAD model was created using CATIA\(^{\ast}\) for a clamshell type ejector nozzle. Design tables allowed linking

\(^{\ast}\) Version 5.18, copyright and registered, Dassault Systèmes, 2008.
parameter definitions within the CAD model to Microsoft Excel\textsuperscript{†} spreadsheets. A schematic representation of the ejector nozzle with clamshell doors is shown in Figure 3.

The closed circuit Boeing wind tunnel at Purdue University’s Aerospace Sciences Laboratory was modified to incorporate the test rig and a second air flow system into the test section to enable testing of the nozzle geometry in desired flow conditions. The test section is 8 ft. in length with a cross-section of 6 ft. x 4 ft. Figure 4 shows a schematic representation of the wind tunnel. The nozzle flow was supplied by a high pressure blower built by the Chicago Blower Corporation. The unit is a centrifugal fan compressor-blower, specifically a Model 2T-30-28 with a 30 hp motor, capable of 2950 cfm at 49.1 in. wg (1.77 psi), giving 2.2 lb/s of mass flow.

The goal in designing the test rig was to keep the exterior surface as free as possible from assembly features, such as bolt holes. It was also necessary to incorporate some flow conditioning, particularly due to the sharp turn the flow has to navigate between the strut and the main body of the test rig, as shown in Figure 5. The flow conditioning was implemented by the application of stainless steel honeycomb and screens, located in between the forward and aft tube bodies. The outer diameter of the rig was 8 in., matching the outer diameter of the test nozzle.

B. Data Acquisition

The data acquisition system used for the quantitative analysis of the flow field consisted of three elements: (1). A 7-hole probe, (2). Pressure measurement and calibration unit, and (3). An automated 2-axis traverse. The primary measurements acquired were wake pressure surveys using a 7-hole probe. The 7-hole probe is a means of tapping seven surface pressures in an unknown flow over a known slender body geometry (the probe). The pressure values and their relations to each other, were used to solve for the flow angles, static pressure, and total pressure of the local flow.

Such wake measurements enable the construction of wake velocity maps as well as thrust analysis via integral control volume thrust calculations. The calibration of the probe used in this work was performed by Jens Lindqvist.\textsuperscript{11} Due to the large number of measurement locations necessary to create an adequate wake survey, an automated 2-axis traverse was employed. The relation of these systems and their setup is depicted in the schematic in Figure 6.

A Pressure Systems Inc., Model 9010 pressure scanner with a static accuracy of ±0.10% was used to efficiently measure the multiple pressure levels tapped by the 7-hole probe. The unit included 16 pressure transducers read by a common analog-to-digital converter. The scanner’s processor used a third order polynomial to convert the pressure transducers’ voltage signal to engineering pressure units. A pitot probe was used to monitor the flow speed in the rig plenum. The values of the rig plenum axial velocity ($U_{PL}$) were used to normalize the wake velocities measured by the 7-hole probe.

C. Flow Visualization

In addition to the velocity survey using the 7-hole probe, different flow visualization techniques were employed to better understand the 3-D structure of the flow. A smoke wand was utilized for preliminary visualization of the flow around the test rig. The smoke was created using kerosene and passed over the ejector nozzle external surface. The surface tufting technique was implemented by fitting the clamshells with tufts of white thread. The thread used was medium duty, sewing thread, composed of a polyester core and a cotton wrapping. These surface tufts align themselves with the local velocity vector showing its direction.

In an effort to obtain a higher spatial resolution visualization of the surface flow, surface sediment tracing was pursued. The surface sediment tracing was achieved by mixing various types of fine powder (sediment) with an evaporative liquid petroleum. As the air flows over the body surface, the suspension aligns itself with the surface flow patterns, and the air gradually evaporates the liquid petroleum. For the work herein, colored chalk powder was used as a suitable sediment with kerosene.

In addition to the above mentioned flow visualization techniques, fluorescent oil flow method was used. The surface oil flow visualization involves placing patterns or smooth coatings of oil on an exposed surface and recording the movement of the oil as it is affected by the flow. The fluorescent oil is excited with ultraviolet light, and filmed through a green-pass optical filter. Using this method, the excitation light is mostly eliminated and significant contrast is captured in the oil patterns. The polyalkylene glycol (PAG) fluorescent oil with a kinematic viscosity of 150 cSt. was used for this application.

IV. Computational Method

A. Computational Grid and Boundary Conditions

As a first step towards the computational analysis of the ejector nozzle, a 3-D grid for the CAD geometry was required. The nozzle and rig surfaces from the CAD geometry were exported in initial graphics exchange specification (IGES) surface format for grid generation. Grids were generated using Pointwise Gridgen version 15.10. Nonoverlapping multiblock hybrid grids were used for this geometry. Because of the symmetry, only one quadrant of the domain was used to save computational cost and time. In the present numerical study, two cases, viz. an ejector nozzle without clamshells (Grid I) and an ejector nozzle with clamshells at 11.5° (Grid II) were considered. Computational meshes for the nozzles are shown in Figure 7. Because of the complex design of the clamshells and the support handle on the nozzle, an unstructured mesh was used near the nozzle and a structured mesh was used in the rest of the flow domain, as shown in Figure 8.

Grid I, corresponding to the ejector nozzle without clamshells, consisted of 20 blocks including one unstructured block near the nozzle throat and resulted in a total of 1.29 million cells. Grid II, corresponding to the ejector nozzle with clamshells, consisted of the same topography with 20 blocks, and a total of 2.53 million cells. The interfaces between structured blocks and the unstructured block consisted of pyramid cells. A variable wall spacing was used for all viscous wall boundaries to yield a $y^+$ value in the range of $30 \sim 300$ and hence wall functions were used for near wall turbulence.

The experimental conditions were low subsonic and hence velocity-based boundary conditions were used in the CFD analysis. The boundary conditions were velocity-inlet at the inflow boundary, inviscid wall along the symmetry planes, velocity-inlet along the outer freestream boundary and pressure-outlet at the exit. Figure 9 shows the numerical values used for various boundary conditions in the CFD simulation of the ejector nozzle with clamshells (Grid II).

B. Computational Solver

The ANSYS FLUENT\textsuperscript{14} version 6.3.26 computational solver was used to solve the Reynolds-averaged Navier-Stokes (RANS) equations and the system of governing equations was closed using turbulence models. FLUENT\textsuperscript{5} is a finite volume numerical solver that can be used with both structured and unstructured meshes. In general, the flow features of high speed jet flows are predicted using density-based numerical approaches. In density-based coupled formulations, the continuity, momentum and energy governing equations are solved simultaneously as a set of equations.

As an alternative, the coupled pressure-based numerical solver\textsuperscript{15} can be used for problems where strong coupling among equations exists. Unlike a segregated algorithm, in which the momentum equation and pressure correction equations are solved one after another in a decoupled manner, a pressure-based coupled algorithm solves a coupled system of equations comprised of the momentum equation and the pressure equation. Since the momentum and pressure equations are solved in a closely coupled manner, the rate of solution convergence significantly improves when compared to the segregated solver. This pressure-based coupled approach can also be used for supersonic and hypersonic problems that cannot be tackled by a segregated approach. Because of these advantages, pressure-based segregated numerical solver was used for all the CFD computations.

C. Turbulence Models and Solver Settings

Menter’s $k$-$\omega$ shear stress transport (SST) turbulence model\textsuperscript{16}, Shih’s realizable $k$-$\epsilon$ turbulence model\textsuperscript{17} with Thies and Tam’s correction\textsuperscript{18} and the standard $k$-$\epsilon$ turbulence model\textsuperscript{19} were used to model the flow turbulence. The $k$-$\omega$ SST turbulence model effectively blends the robustness and accurate formulation of the $k$-$\omega$ model\textsuperscript{20} in the near wall region and the freestream independence of the $k$-$\epsilon$ model\textsuperscript{19} in the far field. Thies and Tam changed some of the coefficients of the standard $k$-$\epsilon$ turbulence model specifically for jet flows and validated the performance of these modified constants with a variety of experimental tests involving round, elliptic and rectangular jets.

For the CFD simulation, a constant density of 1.15 $kg/m^3$ was used. The viscosity was calculated using Sutherland’s three coefficient method. The flow domain was initiated using freestream conditions and the final second order accurate solution was obtained. A Courant-Friedrichs-Lewy (CFL) number of 10 was used.

\textsuperscript{1}Copyright and registered, ANSYS Inc. 2009
for stability reasons. The underrelaxation factors were reduced from their default values to 0.6 (density), 0.6 (turbulent kinetic energy), 0.8 (specific dissipation rate), 0.8 (turbulent viscosity) and 0.6 (energy) for numerical stability. In order to check the convergence, along with the default solver residual monitor, the iteration history of the mass-balance and velocity magnitude at one equivalent diameter downstream of the nozzle throat were monitored. When the mass-balance reached $1 \times 10^{-5}$ and the velocity magnitude reached a steady-state value, the solution was considered as converged.

V. Inlet Distortion

In the experiments, the inlet flow from the blower was passed into the test rig through a flow straightener. This helped in the straightening of the flow and removed the flow nonuniformities. However, some nonuniformity in the velocity distribution was still present. Figure 10 shows the contour of the axial velocity magnitude, measured at the inside of the rig in the absence of the ejector nozzle. It was evident that the experiments involved nonuniform velocity distribution and hence an equivalent uniform velocity inlet was calculated for the CFD simulations using the following procedure.

Table 1. Calculation of the corrected inlet axial velocity magnitude for the CFD simulation of the ejector nozzle without clamshells using the minimization of the RMS difference at $X/D_{EQ}=1.0$.

<table>
<thead>
<tr>
<th>S. No</th>
<th>Corrected inlet velocity ($m/s$)</th>
<th>RMS difference ($Y=0$ plane)</th>
<th>RMS difference ($Z=0$ plane)</th>
<th>RMS average</th>
<th>Variation from experiments (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>44.0</td>
<td>0.0773</td>
<td>0.0873</td>
<td>0.0823</td>
<td>11.13</td>
</tr>
<tr>
<td>2</td>
<td>45.0</td>
<td>0.0819</td>
<td>0.0762</td>
<td>0.0791</td>
<td>09.11</td>
</tr>
<tr>
<td>3</td>
<td>45.5</td>
<td>0.0877</td>
<td>0.0733</td>
<td>0.0805</td>
<td>08.10</td>
</tr>
<tr>
<td>4</td>
<td>46.0</td>
<td>0.0954</td>
<td>0.0722</td>
<td>0.0838</td>
<td>07.09</td>
</tr>
<tr>
<td>5</td>
<td>47.0</td>
<td>0.1148</td>
<td>0.0758</td>
<td>0.0953</td>
<td>05.07</td>
</tr>
<tr>
<td></td>
<td><strong>44.9</strong></td>
<td><strong>0.079</strong></td>
<td><strong>0.079</strong></td>
<td><strong>0.079</strong></td>
<td><strong>09.31</strong></td>
</tr>
</tbody>
</table>

It was necessary to compute the correct mass flow rate through the rig for the accurate comparison between the experiments and computations. The mass flow rate of the nozzle was adjusted in the computation such that it matched the velocity profile at $X/D_{EQ}=1.0$. This corrected mass flow rate gave the magnitude of the velocity inlet used in the CFD simulations. The corrected velocity inlet corresponded to the least root mean square (RMS) difference between the experiments and computations at $X/D_{EQ}=1.0$. The RMS differences were plotted against the inlet velocity magnitude, as shown in Figure 11, and the arithmetic mean of the differences in both $Y=0$ and $Z=0$ planes were used to find the corrected inlet velocity magnitude. Numerical values for the case of the ejector nozzle without clamshells are tabulated in Table 1. It was found that the experimental mass flow (calculated based on the velocity measured by the rig pitot tube) is different from the actual mass flow by 9.31%.

VI. Results and Discussion

The results from the experimental investigation of the ejector nozzle and its numerical simulation are discussed in the following section. The results consist of the qualitative data in the form of flow visualization contours from experiments, velocity contours from experiments and CFD simulations, and comparison of the quantitative data in the form of line plots. The quantitative data presented is normalized using the upstream plenum axial velocity $U_{PL}$, which is the centerline axial velocity in the plenum chamber, upstream of the nozzle for velocity measurements and the equivalent diameter $D_{EQ} = 5.642$ in. of the nozzle throat cross-section for length scale which is defined as:

$$A_t = \pi \left( \frac{D_{EQ}}{2} \right)^2. \tag{1}$$

The origin of the nozzle geometry is shifted to the center of the nozzle throat plane for postprocessing the results.
A. Experimental Results

The experimental results are presented in the form of velocity surveys measured by the 7-hole probe and flow visualization contours. Surveys were conducted on the nozzle with and without clamshells. Conducting surveys on the test nozzle with the clamshells removed had multiple purposes. The core of the primary nozzle’s jet plume was assumed to be free from separation effects, 2-dimensionally symmetric, and in line with the axis of the rig. As such, it served as a suitable datum from which to measure any angular offset of the rig and the 7-hole probe. Finally, it is important to understand the contribution of the primary nozzle’s flow characteristics in order to judge their effects on the ejector nozzle flow field.

Figure 12 shows the wake progression behind the primary nozzle with clamshells, stepping aft through the jet plume. The wake grows increasingly eccentric. This eccentricity follows the divergence of the fixed clamshell support in the vertical direction (along the Z-axis) and the rapid convergence of the primary nozzle in the horizontal direction (along the Y-axis). In the presence of clamshells, a zone of flow separation and recirculation was observed at its inner surface. This is evident from the regions of low velocity at $X = 1.0 D_{EQ}$ in Figure 12.

The transverse flow vectors, constructed from the pressure measurements recorded by the 7-hole probe, are shown in Figure 13. The contours represent the velocity fluctuation intensity. Figure 13(a) shows the velocity fluctuation magnitude for the ejector nozzle without clamshells at $X/D_{EQ} = 1.0$. The velocity fluctuations associated with the ejector nozzle with clamshells at $X/D_{EQ} = 1.0$ are shown in Figure 13(b). The presence of the recirculation zone is confirmed by the region of maximum velocity fluctuations.

In addition to velocity surveys, flow visualization techniques were used for a qualitative study of the flow field. Figure 14 shows the flow visualization phenomenon using the smoke wand. The separation bubble at the ejector slot and reverse flow at the inner surface of the clamshell doors were captured. Figure 15 shows the flow over the nozzle clamshells with surface tufts. It is evident that a recirculation zone is present at the inner surface of the clamshells shown by the orientation of the tufts.

The colored chalk powder distribution obtained using the surface sediment tracing method is shown in Figure 16. The kerosene oil evaporates leaving the sediments on the surface in a fashion defined by the flow field. Figure 16 shows a sample output from the fluorescent technique which corresponds to the ejector nozzle with clamshells at 11.5°. Both surface sediment and fluorescent oil methods confirm the presence of two spiral nodes resulted from the flow recirculation at the inner surface of the clamshells. The reason for this flow separation is, as explained in Section II, the inability of the resulting shear layer to attach to the clamshells’ inner surface.

B. Computational Results

This section presents the results obtained by the CFD simulation of the ejector nozzle without clamshells and the ejector nozzle with clamshells at 11.5° with respect to the support handle. The flow conditions were based on the experimental low speed conditions described in Section IV.A. The computational results are compared with the experiments qualitatively in the form of velocity surveys and quantitatively in the form of jet axial velocity line plots.

1. Ejector Nozzle without Clamshells

The results obtained from the CFD simulation of the ejector nozzle without clamshells are discussed in this section. Without clamshells, the jet behaves like an elliptic jet because of the elliptic cross-section of the nozzle throat. As the nozzle is convergent in nature and the conditions are subsonic, the flow is accelerated to higher velocities. The velocity contour plot of the flow field at the $Z=0$ symmetry plane is shown in Figure 17(a). The CFD simulation of this configuration was necessary for initial comparison with the experimental results before going to the more complicated configuration of a nozzle with clamshells.

Figure 17(b) shows the centerline velocity magnitude profile of jet corresponding to the experiments, Menter’s $k-\omega$ shear stress transport turbulence model, the realizable $k-\epsilon$ turbulence model with Thies and Tam’s model constants and the standard $k-\epsilon$ turbulence model. All the turbulence models used in the present study overpredict the length of the potential core and hence result in an overprediction of the axial velocity magnitude at $X/D_{EQ} = 3$. However, it is observed that $\epsilon$-based turbulence models give a better prediction of the potential core length.

As discussed in Section III, experimental results are available in the form of velocity measurements on Y-Z planes at different $X/D_{EQ}$ locations downstream of the nozzle throat and contour plots of jet cross-
section at these locations. Figure 19 shows the comparison of normalized axial velocity magnitude contours at different axial locations between the experiments and computations. The velocity profiles match very well with the experimental results at $X/D_{EQ}=1$, 1.5 and 2. For the case of $X/D_{EQ}=3$, there is an overprediction of the axial velocity magnitude. This is associated with the inability of the turbulence model to predict the length of the potential core of the jet. At $X/D_{EQ}=3$, as it can be inferred from the experimental results, the potential core ends and the shear layers start to merge with each other. Figure 20 shows the comparison of the normalized axial velocity profile between the experiments and computations.

2. Ejector Nozzle with Clamshells

In addition to the case of the ejector nozzle without clamshells, the CFD simulation of the ejector nozzle with clamshells at 11.5° ejector angle is performed in the present study to complement the experimental findings. Similar to the case without clamshells, experimental results are measured at $X/D_{EQ}=1$, 1.5, 2 and 3. Results obtained from this CFD simulation helps in understanding the flow physics associated with the ejector nozzle and its performance.

The contours of velocity magnitude on the $Z=0$ symmetry plane (ejector plane) are shown in Figure 18. A region of flow separation and recirculation, as found during the experiments, is encountered at the inner surface of the clamshells. Figure 18(a) shows the streamlines and the region of flow separation. The presence of flow recirculation is also evident from Figure 18(b) which shows the negative axial velocity contour lines. This is because of the inability of the resulting free shear layer to skim the inner surface of the clamshells, as discussed in Section II and thereby allow the external atmospheric flow to affect the nozzle exhaust.

Figure 21 shows the comparison of jet cross-section axial velocity contours between the experiments and computations. As with the case without clamshells, similar good agreement at $X/D_{EQ}=1$, 1.5, 2 and over-prediction at 3 is found. The quantitative comparisons of normalized axial velocity in the $Y=0$ and $Z=0$ planes and at different axial locations are shown in Figure 22. Computational results show an overprediction of the potential core length. This results in an increased mass flow at $X/D_{EQ}=3.0$ because of larger entrainment which is evident from the overprediction of the velocity profile compared to the experiments in Figure 22.

VII. Conclusions

A new supersonic jet engine exhaust nozzle system, which consists of clamshell doors as ejectors, was designed and analyzed at experimental conditions. The geometry was parametrically defined using the CATIA CAD package for design exploration. The experimental investigation and CFD simulation of the ejector nozzle with and without clamshells was successfully completed. The velocity profiles at different equivalent diameters downstream of the nozzle throat were measured and compared with the CFD results. Some of the flow physics associated with the ejector nozzle were captured by the application of various flow visualization techniques.

The computational results are in good agreement with the experimental data at axial locations near the nozzle exit and within the jet potential core. An overprediction of the potential core length and axial velocity magnitude at $X/D_{EQ}=3.0$ was observed. Regions of flow separation and recirculation, that limited the performance of the ejector, were encountered near the inner surface of the clamshells. This is because of the inability of the originating shear layer to attach to the inner surface of the clamshells.

The present study concludes the experimental investigation and computation of a supersonic ejector nozzle with clamshells. The flow separation on the inner surface of the clamshells limits the performance of the exhaust nozzle. In future work, the design and analysis of mixing enhancing devices, such as chevrons and tabs in the ejector slot will be investigated to improve performance. This will help in characterizing the benefits of chevrons for the ejector nozzle with clamshells.

Acknowledgments

This work was a part of the supersonic business jet program, sponsored by Rolls-Royce and the Gulfstream Aerospace Corporation. The authors would like to thank Rolls-Royce, Gulfstream Aerospace Corporation and Dynetics for publication approval.
References


15 Lounsbury, R. J., “Pressure Based Coupled Solver in FLUENT 6.3,” 2006 CFD Summit.


Attachment of the free mixing layer with the primary nozzle results in an optimum ejector nozzle performance. The $L/\Delta$ parameter defines the performance of the ejector nozzle.

Operational modes of the ejector nozzle with clamshells (1) Take-off (2) Supersonic cruise, and (3) Approach.

A schematic representation of the ejector nozzle with clamshell doors.
Figure 4. Ejector nozzle test rig in the Boeing Wind Tunnel.

Figure 5. Nozzle rig CAD model and coaxial flow test rig details.

Figure 6. Experimental wake survey data acquisition system and wake survey planes.
Figure 7. Computational mesh a) Ejector nozzle without clamshells, and b) Ejector nozzle with clamshells.

Figure 8. Computational mesh for the quadrant of the ejector nozzle with rig plenum.

Figure 9. Computational boundary conditions for the ejector nozzle with clamshells.
Figure 10. Experimental survey of the rig plenum in the absence of the nozzle showing the nonuniformity involved in the axial velocity magnitude distribution.

Figure 11. RMS difference distribution in the axial velocity magnitude at $X/D_{EQ}=1.0$ downstream of the nozzle throat for the ejector nozzle without clamshells.
Figure 12. Experimental jet wake surveys at a clamshell angle of 11.5°.

Figure 13. Transverse flow jet survey with velocity fluctuation contours at $X/D_{EQ}=1.0$ for: (a) The ejector nozzle without clamshells, and (b) The ejector nozzle with clamshells.
Figure 14. Flow visualization using the kerosene smoke wand showing the separation bubble at the ejector slot and the reverse flow at the clamshell doors.

Figure 15. The experimental rig setup and the flow visualization using surface tufts showing flow reversal.

Figure 16. Flow visualization using the surface sediment method (left) and fluorescent oil (right) showing the spiral nodes.
Figure 17. (a) CFD results at $Z = 0$ plane for the ejector nozzle without clamshells corresponding to the $k-\omega$ SST turbulence model, and (b) Comparison of the centerline axial velocity profiles among experiments, the $k-\omega$ SST, the realizable $k-\epsilon$ and the standard $k-\epsilon$ turbulence models.

Figure 18. Computational axial velocity contours on the $Z = 0$ plane for the ejector nozzle with clamshells showing (a) Streamlines with flow separation, and (b) Reverse flow using negative axial velocity contour lines.
Figure 19. Normalized velocity magnitude contours of experiment and numerical results for ejector nozzle without clamshells.
Figure 20. Normalized axial velocity profiles corresponding to the ejector nozzle without clamshells
Figure 21. Normalized velocity magnitude contours of experimental and numerical results for ejector nozzle with clamshells.
(a) Axial velocity profile corresponding to $X = 1.0 \ D_{EQ}$ and $Z = 0$ plane

(b) Axial velocity profile corresponding to $X = 1.0 \ D_{EQ}$ and $Y = 0$ plane

(c) Axial velocity profile corresponding to $X = 1.5 \ D_{EQ}$ and $Z = 0$ plane

(d) Axial velocity profile corresponding to $X = 1.5 \ D_{EQ}$ and $Y = 0$ plane

(e) Axial velocity profile corresponding to $X = 2.0 \ D_{EQ}$ and $Z = 0$ plane

(f) Axial velocity profile corresponding to $X = 2.0 \ D_{EQ}$ and $Y = 0$ plane

(g) Axial velocity profile corresponding to $X = 3.0 \ D_{EQ}$ and $Z = 0$ plane

(h) Axial velocity profile corresponding to $X = 3.0 \ D_{EQ}$ and $Y = 0$ plane

Figure 22. Normalized axial velocity profiles corresponding to the ejector nozzle with clamshells