CFD Studies Supporting the Design, Testing and Evaluation of Noise Reduction Concepts for Military Aircraft

Sanford M. Dash, Neeraj Sinha, Donald C. Kenzakowski, Chandrasekhar Kannepalli and Ronald J. Ungewitter
Combustion Research and Flow Technology, Inc. (CRAFT Tech)
6210 Keller's Church Road, Pipersville, PA 18947

dash@craft-tech.com

ABSTRACT

This paper provides an overview of CFD studies performed over the past several years which have supported the design, testing, and evaluation of varied jet noise reduction concepts for fighter aircraft. The studies were performed in close collaboration with several University teams that have included Florida State (Krothapalli & coworkers) and the University of Mississippi (Seiner & coworkers), as well as with industry. Concepts evaluated have included chevrons, micro-jets, lobes, and beveling, installed on laboratory models as well as on full-scale engines. Emphasis has focused on concept performance at low altitudes as related to training missions and carrier operations. Scaling up laboratory concepts to full-scale does not account for real engine exhaust complexities associated with internal core/fan mixing (not replicated in the laboratory models), and concepts had to be enhanced (i.e. chevrons made larger) to be effective. Initial studies were for isolated jets exhausting into a quiescent stream with inclusion of lower speed external stream effects found to have a minimal effect on concept effectiveness. However, for fighter aircraft with closely spaced twin nozzles, plume/plume interactions required consideration in evaluating concept effectiveness, as did vehicle/plume interactions. In the most recent studies, CFD was used to evaluate the effectiveness of lobed concepts for a twin nozzle configuration, and varied lobe patterns were also analyzed. Any concept found to be effective for the restricted conditions of interest in these studies (viz., for training missions and carrier landings), may ultimately be installed as a permanent engine upgrade. Hence, additional studies are required to analyze thermal/structural loads on the concepts, their effect on overall performance, and their impact on signatures at higher-altitude operating conditions where noise is not the dominant concern. Upgrades in the CFD are described which improve the ability to analyze the effect of varied concepts with greater accuracy and reliability. These include use of reduced Reynolds-Stress modeling and solving scalar fluctuation equations, with the anisotropy parameters and temperature fluctuations included as inputs to improved analogy-based jet noise prediction codes, providing better agreement with varied data sets. LES studies are supporting the validation and calibration of the upgraded RANS models, but are neither sufficiently mature nor cost effective to be able to analyze realistic exhausts with complex internal core/fan mixing, nor to analyze complex plume/plume and aerodynamic interactions which require grids in excess of 10M cells for RANS simulations.

1.0 INTRODUCTION

The authors, in collaboration with university teams (led by Seiner / Univ. Mississippi and Krothapalli / Florida State), as well as with industry, have used CFD to support the design and evaluation of noise reduction concepts for military aircraft. The work performed has focused on concepts that can be used to reduce jet noise for operational fighter aircraft, with primary noise concerns addressed being those related to low altitude flights over populated areas (as would typically be encountered in training missions), as well as carrier operations.

The multi-faceted role of the CFD studies performed has included:

1. Providing details of the structural changes to the jet flowfields produced by varied concepts, yielding insight into how they work and why they are effective;

2. Defining what specific features should be included in the subscale laboratory tests, such as exhaust geometry details and consideration of dual jet and aerodynamic interaction effects;

3. Delineating the differences in flow structure between the real (full-scale) engine exhaust and the simplified (subscale) laboratory model, and showing how such differences impact concept effectiveness;

4. Performing preliminary screening studies to compare how varied concepts perform, followed by parametric studies to suggest optimal sizes, geometries, and locations;

5. Performing design optimization studies to maximize concept effectiveness; and,

6. Examining how installed noise reduction concepts performs in flight for the low altitude conditions of interest, as well how they effect overall aircraft performance and vulnerability (signature) for operational conditions where noise is not the primary concern.

Varied concepts have been studied over the past several years, such as those shown in Figure 1, with some of the CFD studies performed described in a number of earlier publications [1][2]. Subscale testing has primarily provided noise data [3][4] so there is not a one-to-one correspondence between the data and the CFD results. However, the relative effectiveness of varied concepts based on the CFD calculations (and on noise-related estimates based on the CFD) have been in direct accord with the acoustic data obtained.

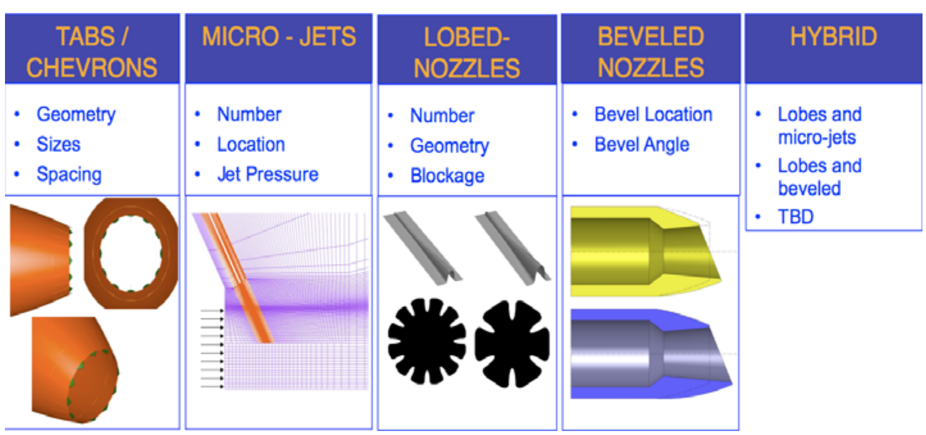


Figure 1. Preliminary Noise Reduction Concepts Studied.

In this paper we will focus on how CFD has been used to support design, testing, and concept evaluation rather than on the effectiveness of the varied concepts studied. In Section 2, we discuss the CFD methodology and turbulence modelling utilized, the building-block validation process implemented, and how we have used

CFD in unison with testing in a synergistic manner. Section 3 provides a brief overview of the CFD noise reduction concept studies performed and the insights that have been achieved as a result of these studies. In Section 4, we discuss recent progress made in improving the CFD utilized for jet noise reduction research.

2.0 CFD METHODOLOGY AND ITS UTILIZATION

To support jet noise reduction concept design, screening, and evaluation studies, we have primarily used RANS–based CFD methodology in conjunction with testing, following the steps listed in Table 1. LES methodology, requiring significantly larger CPU resources, is used for validation of the RANS turbulence models that are utilized. Isolated (single nozzle) jet CFD studies are performed first (Step 1), using a simplified model of the exhaust (without complex internal core/fan mixing details). Varied noise reduction concepts are screened to see how they alter the mean flow structural features (i.e., shock pattern) and turbulence. A well-established structured grid CFD code [1][2] with a turbulence model validated for high-speed aero-propulsive flows [5] has proven sufficient for these screening studies, as well as for the parametric studies that follow where we examine the concept sizing, geometry, and placement that is most effective. The use of “refined” structured grids that adequately resolve the basic jet mixing layer, as well as the smaller scale structures produced by the noise reduction concepts is critical in concept screening and evaluation. The evaluation of how effective these concepts are in reducing noise requires the use of jet noise codes, such as those based on analogy methods [6][7], but even simpler techniques, such as comparing the changes to the integral turbulent kinetic energy variation and/or the jet shock structure have proven useful in comparing the effectiveness of one concept to another.

Table 1. Step-by-Step Process Implemented for Design and Evaluation of Jet Noise Reduction Concepts.

STEPS	CFD	TESTS
Step 1 Isolated Lab Scale Studies	• Concept Design, Screening and Parametric Studies - Still air and external stream	• Tests of Concepts shown to be effective in CFD Studies
Step 2 Interaction Studies	• Evaluation of Plume/Plume and Aerodynamic Interactions on Concept Performance	• Non-Isolated Lab Scale Tests - Dual plumes - Aero effects
Step 3 Real Engine Exhaust Studies	• Comparison of Subscale Model Vs. Real Exhaust (with Internal Core/Fan Mixing) • Concept Design Mods for Full-Scale	• Real engine testing on test stand with and without concepts installed
Step 4 Full Vehicle in Flight	• Studies at Lower Altitudes - Concept Performance • Studies at all Altitudes - Overall Performance	• Flight Tests

In Step 2, CFD studies are performed to examine how plume/plume and/or aerodynamic interactions impact the evaluation of concept effectiveness, which provides guidance for the subscale tests to be performed. Such studies have often indicated the need to repeat isolated (single nozzle) laboratory tests with dual nozzle tests (as would be needed for fighter aircraft with closely spaced nozzles), and sometimes with tests that introduce the effects of vehicle aerodynamic interactions with the plume in a simplified manner.

Figure 2 shows predicted turbulent kinetic energy (TKE) contours for a dual nozzle fighter aircraft at several axial stations downstream of the nozzle exit plane performed in an isolated manner (single plume), with plume/plume interactions, and with aerodynamic interaction effects accounted for. Due to a slight inward canting of the nozzles, as well to entrainment effects, the plume/plume interactions lead to an earlier merger of the two plumes with somewhat faster mixing. With aerodynamic interactions accounted for (requiring a full vehicle solution), peak TKE levels are reduced with mixing modified in part due to vehicle boundary layer effects, but also due to flow non-uniformities in the region between the plumes. The analysis of the complete vehicle with plumes [8] requires the use of a multi-element unstructured code and implementation of specialized grid topology, with hex elements used in the plume mixing region for accuracy, as shown in Figure 3.

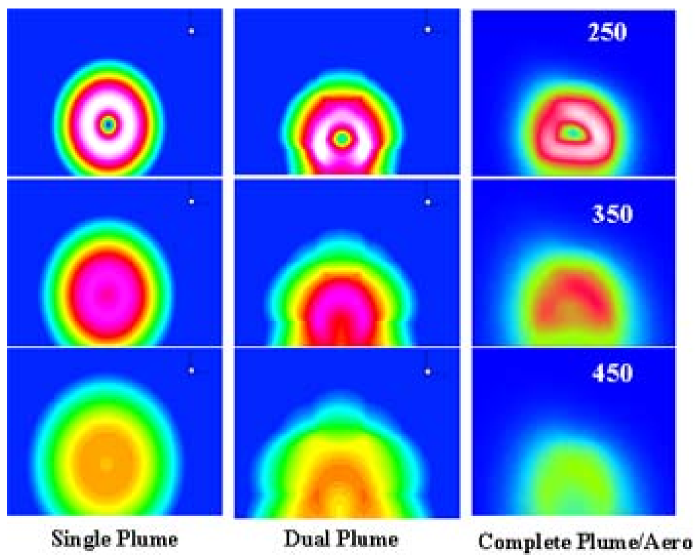


Figure 2. Comparisons of Single & Dual Plume Solutions, and, Solution with Aerodynamic Interactions for Laboratory Configuration with Closely Spaced Nozzles.

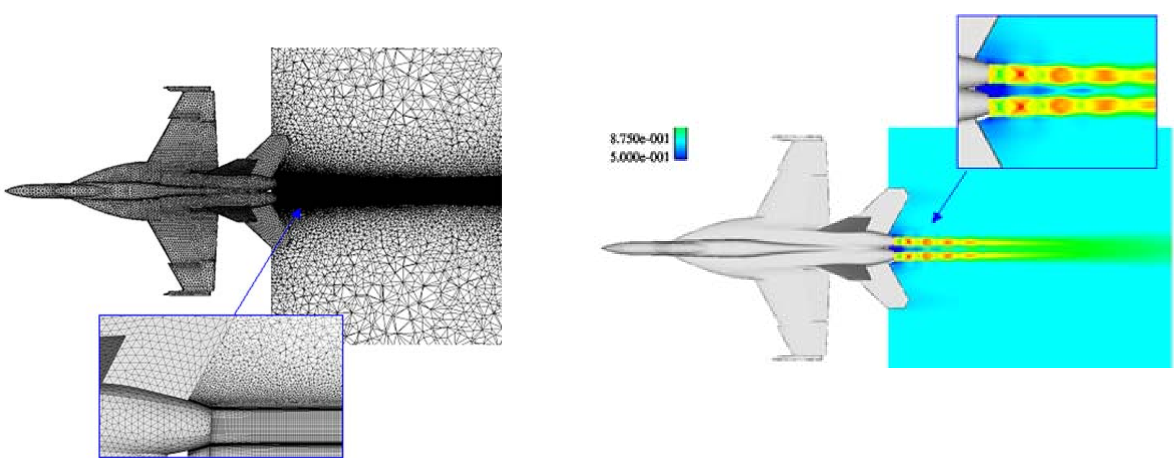


Figure 3. Specialized Multi-Element Grid and Predicted Plume Structure for Dual Nozzle Fighter Aircraft Flowfield.

In Step 3, we use CFD to study differences in flow characteristics between the simplified laboratory model and the real engine [9]. In modern military engines, internal core/fan mixing features can be extremely complex and they are not easily reproducible in subscale laboratory models. By-pass flow may be introduced through a series of slots and vents to cool the aft liner surface (and thus minimize hot part radiation), which leads to a thick wall layer of slower speed, cooler flow that is not replicated in the subscale models (where the core and the fan flows are pre-mixed). Figure 4 exhibits such differences showing comparisons of Mach number and turbulent kinetic energy for a subscale laboratory engine (with fully mixed core/fan flow) and a real engine (with distributed bypass flow). Note the thick wall layer in the real engine as clearly seen in the Mach number contours. Noise reduction concepts designed and tested for the simplified subscale model, not only have to be scaled up for the real engine, but often have to be modified to be equally effective. In our chevron studies, effectiveness required that they extend through the near wall layer and into the higher speed core flow for the real engine and thus, they had to be larger than chevrons that were simply geometrically scaled-up. For lobed (corrugated) nozzle concepts, the exit plane conditions for the subscale lab model with premixed core/fan flow, and the real engine are markedly different, as shown in Figure 5, which must be accounted for in the design process.

The Step 4 studies have examined how effective the noise reduction concepts are for flight conditions where there are noise concerns, as well how they affect overall performance and vehicle vulnerability for all operating conditions. Such studies introduce the complexities of vehicle installation and aerodynamic interaction effects [10][11], which can have a significant influence on the noise produced, and thus can indicate the modifications to the concept design and placement that are required. Concepts designed to reduce noise at lower altitudes may have an adverse effect on vehicle performance and vulnerability at higher-altitudes, and compromises in the design may be required. In addition, concept survivability concerns must be addressed, so that issues such as thermal loads (including afterburning effects) and design extensions to deal with such loads [12] come into play in the evaluation process.

Substantive model validation has been performed to ensure that the CFD solutions are adequate to support design and evaluation studies. A building-block approach [13], formulated quite a number of years ago, has been followed in our validation work. In this approach, jet solutions for problems of increasing complexity are compared with data to calibrate the RANS turbulence model for problems encompassing the environment of interest. For jets and free shear flows, the high-Re turbulence model coefficients are calibrated with the inclusion of extensions for high-speed compressibility effects and vortex stretching, and for analyzing multiple-cell shock structure [14]. A unified k-epsilon turbulence model with invariant coefficients [5] was found to be adequate for analyzing a broad variety of jet flowfields.

The analysis of wall bounded flows, such as internal core/fan mixing processes, requires the extension of this model to include low Re, near-wall damping terms. Turbulence models specialized for boundary layer flows do not do a good job for jets, and visa versa. In earlier work, we utilized a hybrid approach employing the So, Sarkar, Gerodimos, & Zhang (SSGZ) low Re model [15] in near wall regions and the high Re model away from the wall, as described in Ref. [5]. This proved quite cumbersome for complex flows so we recalibrated the SSGZ model using the high Re coefficients via modifying the near-wall damping term coefficients [16]. This improved unified model works well for both free jets and wall bounded mixing problems, and has been successfully used for analyzing a broad variety of aero-propulsive problems. Recent scalar transport [17] and explicit algebraic stress [18] extensions and their role in improving predictions for jet noise concept evaluation [19][20] will be summarized in Section 4.

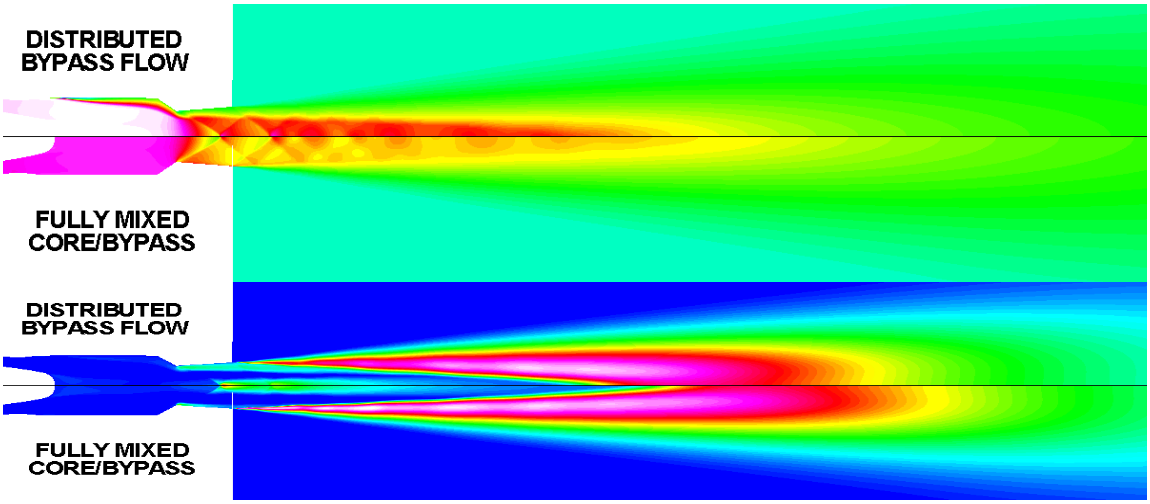


Figure 4. Comparison of Mach number (upper fig.) and TKE contours (lower fig.) for real engine with distributed exhaust and subscale model with premixed core/bypass flow.

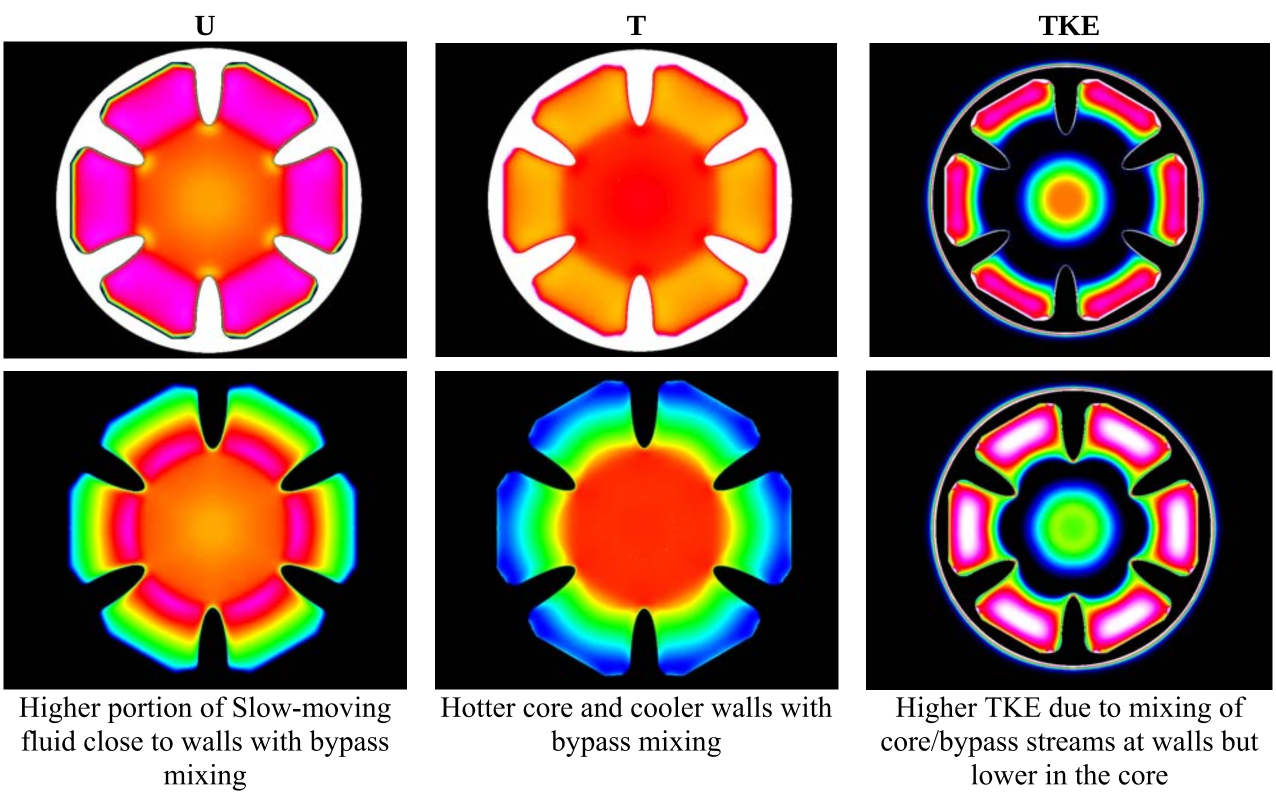
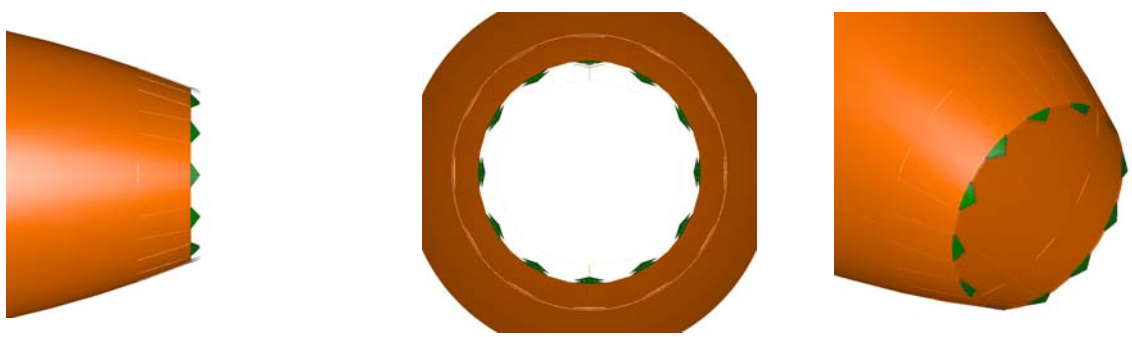


Figure 5. Nozzle Exit Plane Contours with Lobed Noise Reduction Concept Showing Differences Between Subscale Lab Model with Premixed Core/Fan Flow (Upper Fig.) and Real Exhaust (Lower Fig.) with Distributed By-Pass Flow.

3.0 CONCEPT DESIGN AND EVALUATION STUDIES

The studies performed in designing and evaluating noise reduction concepts, such as those shown in Figure 1, have followed the multi-step approach of Table 1, with the role of CFD to be highlighted in the discussion that follows. For lab scale studies with simplified exhaust models, the CFD work was performed in unison with experiments [3][4]. For full-scale (test-stand) studies on real engines, only a few of the concepts studied were tested. Full vehicle studies were largely exploratory and not accompanied by flight tests. Key data to compare with CFD for flight tests would be that of IR imagery, which has been done in programs not related to jet noise reduction.

In our initial fighter aircraft noise reduction concept studies, we examined the utilization of **conventional chevrons,** which generate streamwise vortical enhancements to the mixing, and had proven effective in reducing jet noise for subsonic, separate flow nozzles [21]. For supersonic exhausts of interest, we varied the chevron sizes, their number and spacing, and their inclination into the exhaust. One of the configurations studied is shown in Figure 6, where 12 chevrons are symmetrically arranged. The CFD studies for a laboratory model with pre-mixed core flow [1][2] indicated that use of a smaller number of chevrons was more effective (i.e., 6 worked better than 12), but that the sizes and inclination into the exhaust that were needed to provide significant noise reduction, also produced significant thrust losses. These observations were confirmed by experiments [22]. For the real engine, simply scaling up the chevron size was not adequate since to be effective, the chevrons had to be large enough to penetrate through the slower speed by-pass flow wall layer (Figure 4) into the higher speed core flow. These larger chevrons produced more substantive thrust losses with the studies performed clearly showing that concepts designed for an idealized exhaust may require substantive modifications for the real engine.



Side view Front view

Figure 6. 12 chevrons arranged symmetrically in 30 deg. Intervals.

Micro-jets, in some senses act like a fluidic chevron, but liquid (water) jets introduce multi-phase processes (droplet breakup, inter-phase drag, etc.) that also modify the noise produced by the jet [23]. We had performed CFD studies comparable to those performed for chevrons, where the number of jets, their spacing and orientation, and the jet pressure ratio were varied [1][2] to support laboratory studies. Our studies, restricted to air jets, provided guidance to the laboratory experiments and showed this concept to be effective in attenuating the nearfield turbulent kinetic energy, which resulted in noise reduction. A CFD grid in the plane of one of the micro-jet injectors is shown in Figure 7. Note the high resolution used for the nearfield micro-jet flow structure, needed to accurately predict the penetration and mixing. The complexities in modelling liquid micro-jet injection precluded us from providing CFD support for such studies when the experiments were being performed. However, recent advances made for analyzing liquid jets injected into high-speed streams [24] would permit us to now perform such calculations with some degree of reliability.

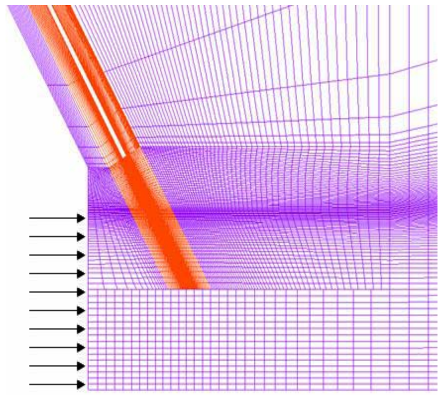


Figure 7. CFD Grid in Plane of Micro-Jet Injector.

Substantive CFD work was performed in supporting the design and evaluation of **lobed concepts**. With this concept, mixing is enhanced via vortical structures generated by the lobes, without the blockage losses produced by chevrons. The desired area variation modifications to the baseline geometry, to be provided by the lobes, is determined by the method of characteristic, but the lobe height, width and spacing (number of lobes) is determined via the performance of parametric CFD studies. Figure 8 shows several 12 lobe and 6 lobe configurations. The "a" configurations are broader, while the "b" configurations penetrate more into the flow. The lobes initiate near the nozzle throat and extend to the trailing edge. Figure 9 shows predicted turbulent kinetic energy (TKE) contours in symmetry planes passing through the lobes. It is seen that the 6 lobe configuration is more effective than the 12 lobe configuration. With 12 lobes, the vortices are too closely spaced, interacting and losing their effectiveness a short distance downstream of the nozzle exit plane. The broader lobes of "6a" are somewhat more effective than the "6b" lobes. A comparison of integral TKE contours for the baseline (no lobe) and "6a" lobe configurations is shown in Figure 10 which can be qualitatively related to the noise reduction levels achieved, as ascertained by the experiments performed [4][22]. Reducing the number of lobes to less than 6 proved to be less effective, so that 6 lobes was indeed optimal.

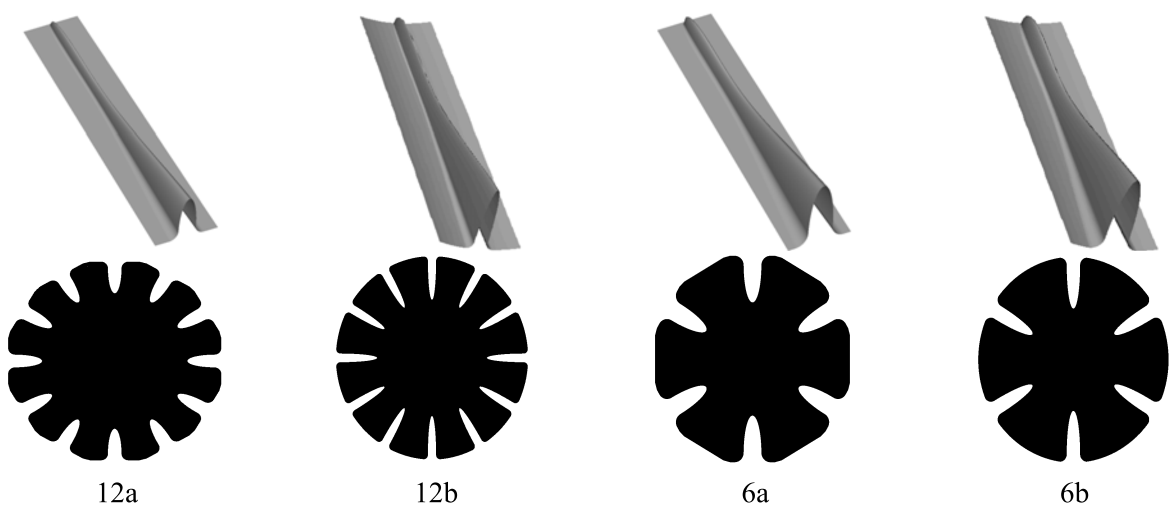
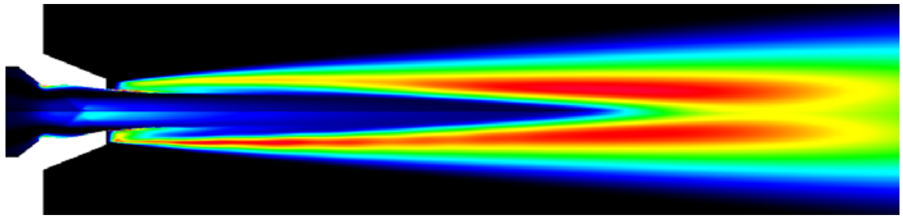
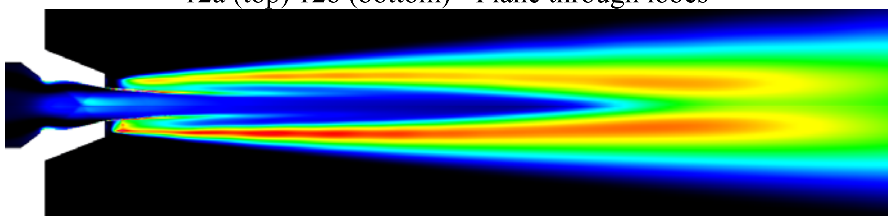


Figure 8. Lobe Configurations for 1/10th Laboratory Model Nozzle.



12a (top) 12b (bottom) - Plane through lobes



6a (top), 6b (bottom) - Plane through lobes

Figure 9. TKE Contours in Symmetry Plane Through Lobes.

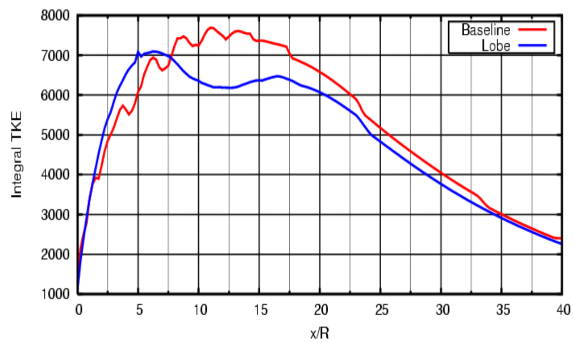


Figure 10. Integral TKE Distribution Along Axial Direction.

Both laboratory subscale tests and real engine tests were performed with the above 6 lobe configuration, with the installed lobes shown for both sets of tests in Figure 11 and Figure 12. For the full-scale engine, the lobes were simply scaled-up from those of the subscale model. In view of the marked differences in exit plane properties for pre-mixed and non-premixed conditions, as shown in Figure 5 for the 6 lobe configuration, the design for the real engine could have been further improved, although it was found to provide noise reduction levels comparable to those for the laboratory model [4][22]. With dual, closely spaced and canted inward nozzles, plume/plume interactions may impact the effectiveness of concept designs based on isolated studies, such as those described above. For the chevron, micro-jet, and lobed concepts, such interactions do play a role, but they do not substantially alter the overall concept effectiveness, as ascertained by the performance of dual nozzle CFD studies, as well as by dual-nozzle testing. CFD solutions for an isolated, single nozzle can make use of lobe symmetry (i.e., pie slice grid with symmetry planes bisecting and between lobes is used), but for dual nozzle calculations (Figure 13), the full nozzle must be analyzed (since only half-plane symmetry in the plane between the nozzles is available). The multi-element unstructured grid for a 6 lobe configuration is shown in Figure 14, with predicted contours of Mach number and TKE shown in Figure 15 and Figure 16.

Figure 11. 1/10th Scale Model Nozzle with lobes installed.

Figure 12. Gas Turbine Engine On Test Stand With Lobes (Corrugated Engine Seals).

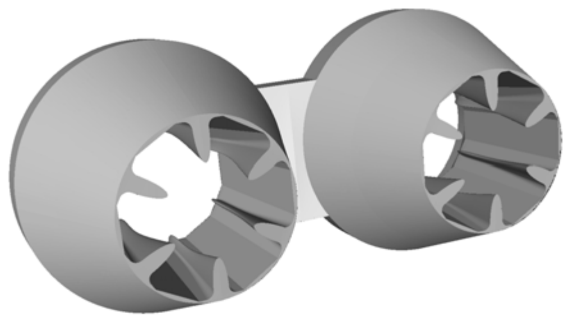


Figure 13. Dual Nozzle Geometry with 6 Lobes

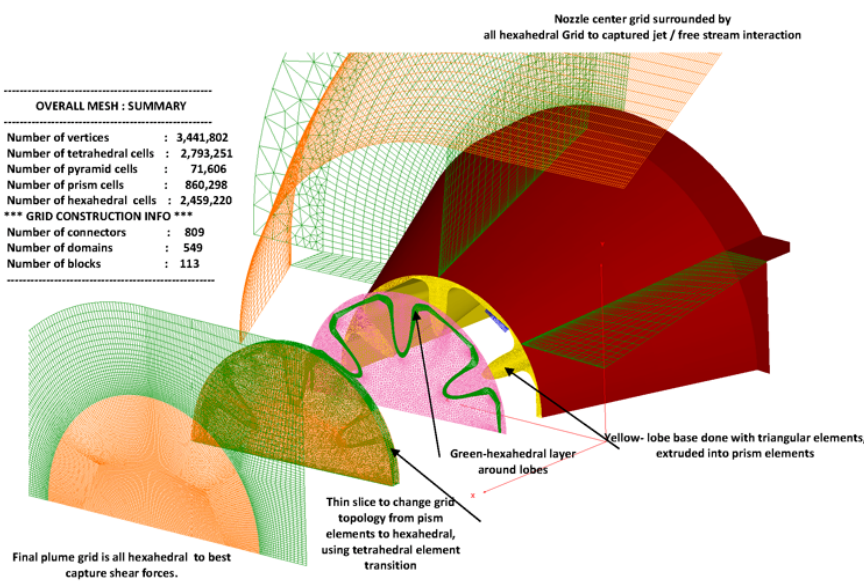


Figure 14. Multi-element Unstructured Grid for Nozzle with 6 Lobes.

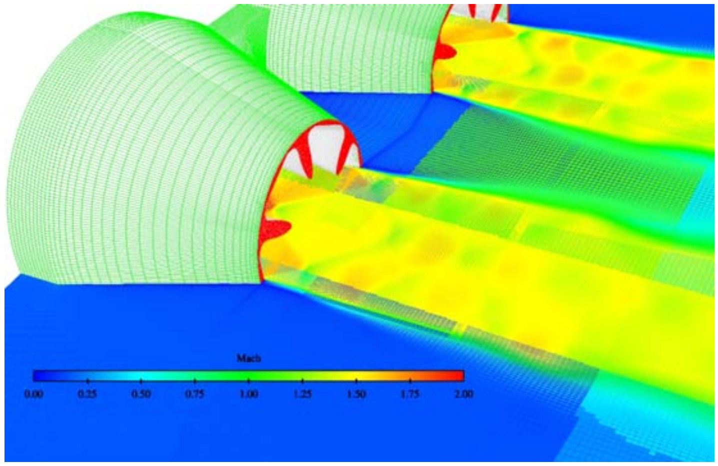


Figure 15. Mach Number Contours Superimposed on Multi-element Unstructured Grid for Dual Nozzle Study with 6 Lobes.

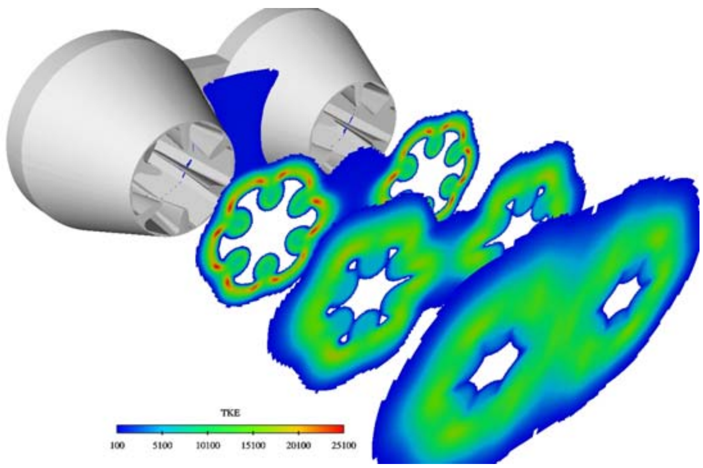


Figure 16. TKE Contours for 6-Lobe Nozzle Dual Plume Calculation.

With **bevelled concepts**, dual nozzle calculations are always required since bevelling modifies the plume expansion and changes how and where the plumes interact. We have examined the impact of the bevel angle, and have also investigated differences between vertical and horizontal bevelling (Figure 17). Contours of TKE and temperature are shown in Figure 18 comparing the solutions for a 24 degree bevelled angle case, which has proven to be a most effective angle to use. The horizontal configuration provides somewhat better noise reduction, as indicated in the integral TKE comparison of Figure 19, and it does not alter vehicle moments, which vertical bevelling does. However, with bevels there is directionality of the noise, with the greatest reduction occurring in the direction of the elongated bevelling, so that for the flyover concerns of interest, vertical bevelling would be more effective.

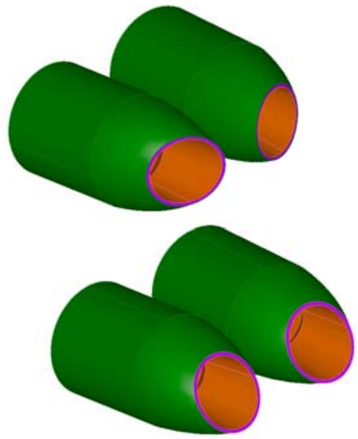


Figure 17. Horizontal (upper) and Vertical (lower) Beveled Configurations.

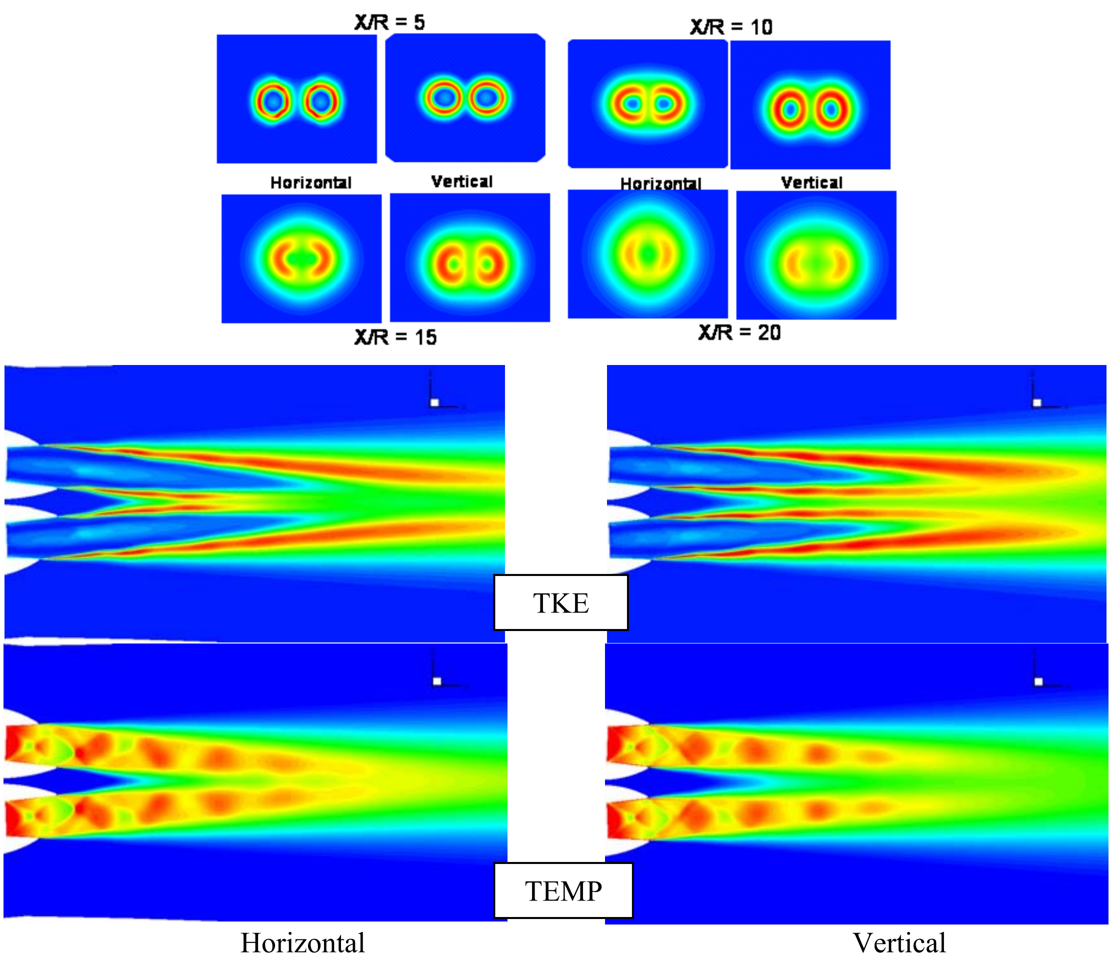


Figure 18. Comparison of Horizontal and Vertical Dual, 24 Degree Beveled Nozzle Solutions.

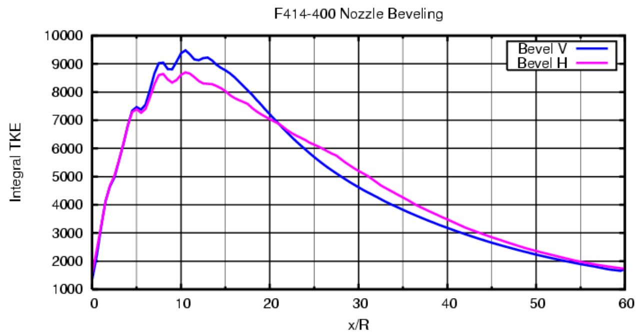


Figure 19. Integral TKE Comparison of Horizontal and Vertical Dual Nozzle, 24 Degree Beveled Configurations.

A variety of hybrid concepts had also been investigated with CFD which combine some of the concepts discussed above. One of the hybrid concepts looked at was the use of beveling with lobes, where the two concepts combined can improve upon the use of either concept by itself. Figure 20 shows contours of temperature and TKE in a horizontal symmetry for this hybrid concept, with TKE comparisons showing reduced levels in comparison to those for the lobe-only or beveled-only solutions. Other hybrid concepts have included those of lobes with chevrons and lobes with fluidic concepts (see Ref. [12]).

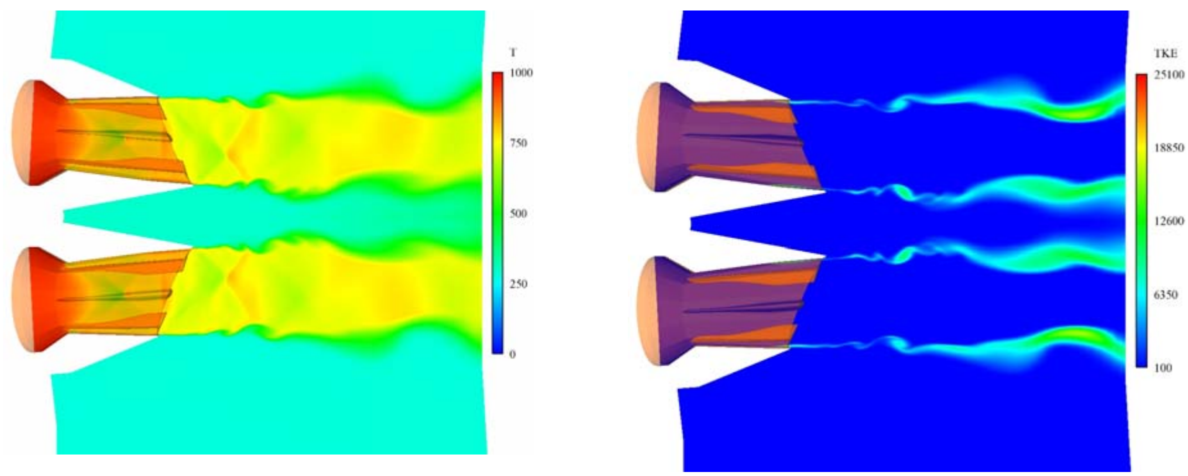


Figure 20. Lobed Nozzle Result of Temperature and TKE at Horizontal Symmetry Plane.

4.0 RECENT UPGRADES TO THE CFD METHODOLOGY UTILIZED

CFD upgrades which are improving the accuracy of the jet calculations performed have focused on RANS turbulence modelling extensions to incorporate the effects of scalar fluctuations and anisotropy on the flow structure. Scalar fluctuation model (SFM) work was initiated nearly a decade ago with attention then focused on Prandtl number variations in hot, subsonic jets [25]. The inclusion of compressibility effects for supersonic jets [17] and combustion effects [26] for reacting flows has entailed substantive model upgrades and the performance of systematic validation studies. The SFM extends conventional two-equation ($k\varepsilon$) turbulence models by solving additional equations for energy and species fluctuations (k_E, k_α) and their dissipation rates (ε_E, ε_α). These variables are then used to provide variations in the turbulent Prandtl and Schmidt numbers using local time-scale relations. For simple flows, such as that of a hot round jet into still air, an average value of Prandtl number may suffice, but what value to use for different conditions is not know apriori. Using SFM, we have generally obtained results consistent with data for basic flows, as exemplified by comparisons of solutions for Seiner's hot, Mach 2 jet experiment using the compressibility-corrected kε model of Ref. [5] with several prescrived values of turbulent Prandtl number (0.6 is seen to work best), and , using the SFM (Figure 21). We also show predicted Prandtl number variations for this case, which are seen to an average level of about .6, but vary from .4 to .8 across the jet.

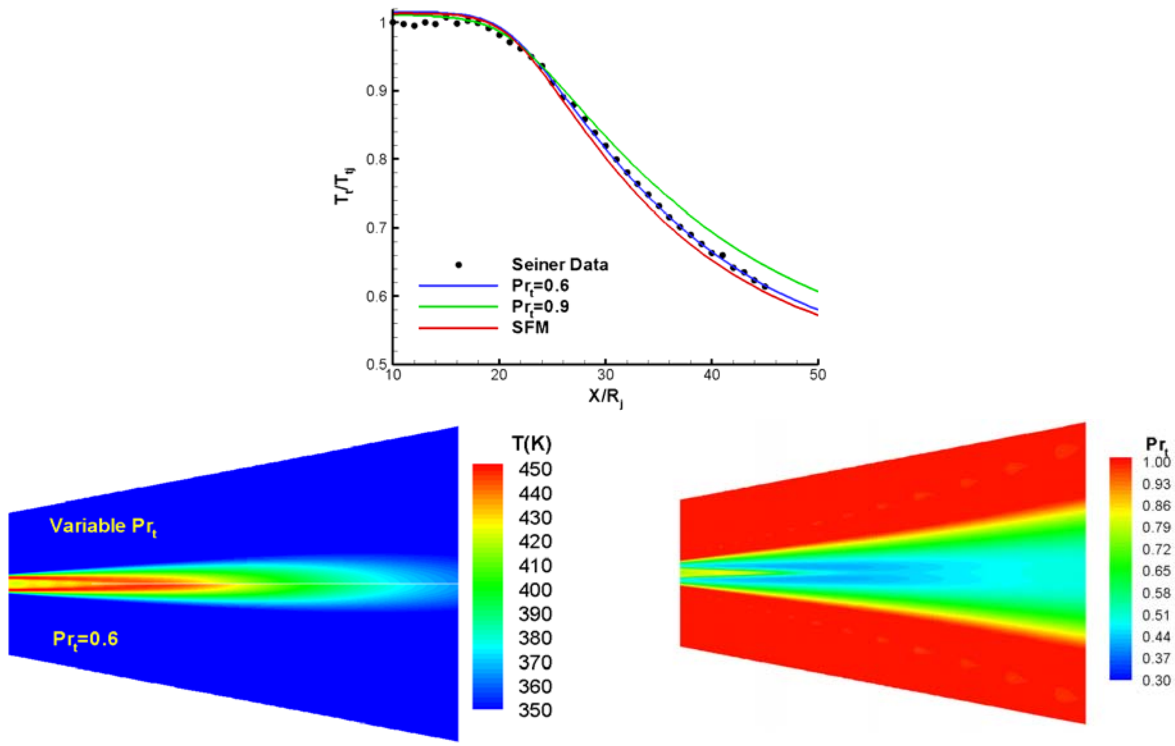


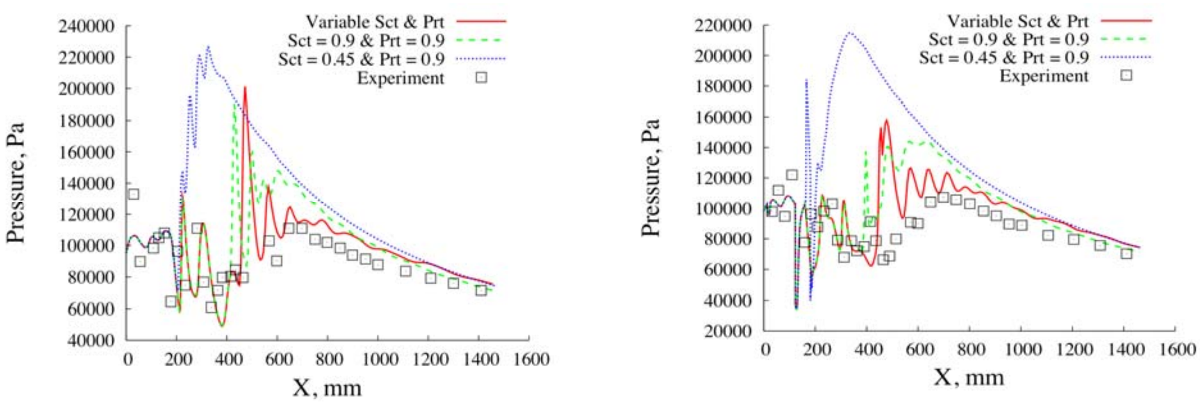
Figure 21. Analysis of Seiner Mach 2 Hot Jet.

At the other extreme, where consider a much more complex 3D flow (with or without combustion), variations in Prandtl and Schmidt number can be very substantive and no single values will suffice. Consider the SCHOLAR data entailing the mixing and burning of hydrogen injected into a high speed ducted air stream. Calculations by a number of investigators have clearly demonstrated the inability to match this data using constant values of Prandtl and Schmidt number, no matter what combinations were chosen. This is illustrated in Figure 22 showing comparisons with data using constant values and the improved comparisons obtained using the SFM (see Ref. [27] for details).

Validation of the SFM is being accomplished using a new Building Block Data Base (BBDB) tool [28] where we are storing archival data sets, accompanying LES solutions, and the RANS solutions performed using the SFM. The BBDB is web-based and GUI driven, and, is intended to be used for CFD code validation for high speed mixing problems by varied NASA/DoD and University groups. Key to the detailed validation of advanced RANS turbulence models is the need for velocity fluctuation data, as well as scalar fluctuation data which is extremely difficult to obtain for jet/propulsive environments of interest. The multi-step approach schematized in Figure 23 is being utilized where PIV data is used to establish the validity of LES solutions (for velocity fluctuations) and the LES solution is used to provide the scalar fluctuation statistics. The scalar fluctuations are not only used to generate variable Prandtl and Schmidt numbers, but they are used to improve jet noise predictions using analogy-based codes and are also used to provide the temperature/species variances needed for assumed PDF-based turbulent combustion models. Improvements in jet noise predictions via the inclusion of temperature fluctuation effects into the JENO code are described in Ref. [6][20].

Our work with Explicit Algebraic Stress Models (EASM) for jets has required modifications to established EASM models to incorporate compressibility effects and to specialize these models for jet flows, resulting in a specialized EASM/J variant described in Refs. [18][19]. In addition to providing some improvements in the overall flow predictions, the local anisotropy of the turbulence is predicted which has been found to provide improvements in making jet noise predictions (see Ref. [20]).

The jet noise codes we have worked with [6][7] have often provided adequate predictions for simple, round jets, but their ability to provide reasonable estimates for the changes in jet noise associated with implementation of varied noise reduction concepts has been very limited. At this time, estimates from these codes have not proven reliable. We have been working with the developers (Khavaran [6]) and Hunter and Thomas [7] to try and improve their overall capabilities, so that these codes can provide more reliable estimates.



(a) lower wall (b) upper wall

Figure 22. SCHOLAR Combustion Experiment, Wall Pressure Predictions Vs. Data.

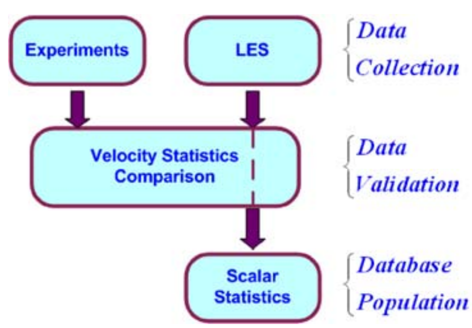


Figure 23. Unified Experimental and LES Framework to Obtain Turbulence Data for Calibration and Validation of Scalar Fluctuation Models.

5.0 ACKNOWLEDGEMENTS

This research was supported by the Office of Naval Research and monitored by Dr. Gabriel Roy.

6.0 REFERENCES

[1] Dash, S.M., Kenzakowski, D.C., and Kannepalli, C., "CFD Simulations of Supersonic Jet Noise Reduction Concepts for Application to Military Aircraft," Advances in Combustion and Noise Control, Cranfield University Press, Ed. G.D. Roy, K.H. Yu, J.H. Whitelaw, J.J. Whitton, pp. 303-324, 2005.

[2] Dash, S.M., Kenzakowski, D.C., Sinha, N., and Seiner, J.M., "Mixing Enhancement Concepts for Fighter Aircraft for IR/Noise Suppression," MSS Specialty Group on Infrared Countermeasures, Signatures and Suppression Session, Monterey, CA, 24-27 April 2006.

[3] Krothapalli, A., "High-Speed Jet Noise Advances in Combustion and Noise Control, Cranfield University Press, Ed. G.D. Roy, K.H. Yu, J.H. Whitelaw, J.J. Whitton, pp. 259-286, 2005.

[4] Seiner, J.M., Ukeiley, L., Jansen, B., Kannepalli, C., and Dash, S.M., "Noise Reduction Technology for F/A-18E/F Aircraft," AIAA-2004-2972, 10th AIAA/CEAS Aeroacoustics Conference & Exhibit, Manchester, UK, 10-12 May 2004.

[5] Papp, J.L. and Dash, S.M., "Turbulence Model Unification and Assessment for High-Speed Aeropropulsive Flows," AIAA Paper No. 2001-0880, 39th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 8-11, 2001.

[6] Khavaran, A., and Kenzakowski, D.C., "Progress Toward Improving Jet Noise Predictions in Hot Jets", AIAA Paper 2007-0012, 2007.

[7] Hunter, C.A. and Thomas R.H., "Development of a Jet Noise Prediction Method for Installed Jet Configurations," AIAA Paper 2003-3169, 2003.

[8] Shipman, J.D., Cavallo, P.A., and Hosangadi, A., "Efficient Simulation of Aircraft Exhaust Plume Flows Using a Multi-Element Unstructured Methodology," AIAA-2001-0598, 39th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 8-11, 2001.

[9] Dash, S.M., "Simulation of Laboratory Jets and Full Scale Exhaust Plumes," INVITED PAPER, Paper No. FEDSM2003-45056, 4th ASME_JSME Joint Fluids Engineering Conference, Honolulu, HI, July 6-11, 2003

[10] Seiner, J.M., Ponton, M.K., Jansen, B.J., Dash, S.M., and Kenzakowski, D.C., "Installation Effects on High Speed Plume Evolution," Paper No. FEDSM97-3227, 1997 ASME Fluids Engineering Division Summer Meeting, Vancouver, British Columbia, Canada, June 22–26, 1997.

[11] Dash, S.M., Kenzakowski, D.C., and Sinha, N., "Turbulent Aspects of Plume Aerodynamic Interactions," AIAA-95-2373, 31st AIAA Joint Propulsion Conference, San Diego, CA, July 10-12, 1995.

[12] Sinha, N., Kenzakowski, D.C., Ungewitter, R.J., Dash, S.M., and Seiner, J.M., "Computational and Experimental Investigations of Jet Noise Reduction Concepts for Low-Bypass Ratio Military Gas Turbine Engines" Proceedings of ASME Turbo Expo 2008, Power for Land, Sea and Air, Berlin Germany, ASME Paper No. GT2008-50091, June 9-13, 2008.

[13] Dash, S.M., Walker, B.J., and Seiner, J.M., "A Building-Block Approach for Turbulence Model Validation/Upgrade Focused on High-Speed Flows," 19th JANNAF Propulsion Meeting, Feb. 24-27, 1992.

[14] Dash, S.M., Kenzakowski, D.C., and Papp, J., "Progress In Jet Turbulence Modeling For Aero-Acoustic

Applications," Paper No. AIAA-2002-2316, 8th AIAA/CEAS Aeroacoustics Meeting, Breckinridge, CO, June 17-19, 2002.

[15] So. R. M. C., Sarkar, S., Gerodimos, G., and Zhang J., "A Dissipation Rate Equation for Low-Reynolds-Number and Near-Wall Turbulence", *Theoretical Computational Fluid Dynamics*, Vol. 9, pp. 47-63.

[16] Papp, J.L. and Dash, S.M., "A Rapid Engineering Approach To Modeling Hypersonic Laminar To Turbulent Transitional Flows," *Journal of Spacecraft and Rockets,* Vol. 42, No. 3, pp. 467-475, May-June, 2005.

[17] Brinckman, K. W., Calhoon, W.H., Jr., and Dash, S.M., "Scalar Fluctuation Modeling for High-Speed Aeropropulsive Flows," *AIAA Journal*, Vol. 45, No. 5, pp 1036-1046, May 2007.

[18] Papp, J.L., Kenzakowski, D.C., and Dash, S.M., "Calibration and Validation of EASM Turbulence Model for Jet Flowfields," AIAA Paper No. AIAA-2002-0855, 40th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 14-17, 2002.

[19] Kenzakowski, D.C., and Papp, J.L., "EASM/J Extensions and Evaluation for Jet Noise Prediction," AIAA Paper No. AIAA-2005-0419, 43rd Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 10-13, 2005.

[20] Kenzakowski, D.C., and Kannepalli, C., "Jet Simulation for Noise Prediction Using Advanced Turbulence Modeling," AIAA Paper No. AIAA-2005-3086, 11th Aeroacoustics Conference, Monterey, CA, May 23-26, 2005.

[21] Kenzakowski, D.C., and Dash, S.M., " Study Of Three-Stream Laboratory Jets With Passive Mixing Enhancements For Noise Reduction," AIAA Paper No. 2000-0219, 38th AIAA Aerospace Sciences Meeting at Reno, NV, Jan 10-13, 2000.

[22] Seiner, J.M., Jansen, B.J., Ukeiley, L., "Military Aircraft Noise Reduction Technology for the FCLP Mission," Advances in Combustion and Noise Control, Cranfield University Press, Ed. G.D. Roy, K.H. Yu, J.H. Whitelaw, J.J. Whitton, pp. 287-302, 2005.

[23] Krothapalli, A., Venkatakrishnan, L., Lourenco, L., Greska, B., and Elavarasan, R., "Turbulence and Noise Suppression of a High-Speed Jet by Water Injection, *J. Fluid Mech*., Vol. 491, pp. 131-159, 2003.

[24] Brinckman, K., Hosangadi, A., Ahuja, V., Dash, S.M. and Feldman, G., "A CFD Methodology for Liquid Jet Breakup and Vaporization Predictions in Compressible Flows," 46th Aerospace Sciences Meeting and Exhibit, AIAA Paper No. AIAA-2008-1023, Reno, NV, 7-10 Jan 2008.

[25] Chidambaram, N., Dash, S.M., and Kenzakowski, D.C., "Scalar Variance Transport in the Turbulence Modeling of Propulsive Jets," *Journal of Propulsion and Power*, Jan-Feb., 2001.

[26] Calhoon, W.H., Jr., Brinckman, K.W., Tomes, J., Mattick, S., and Dash, S.M., "Scalar Fluctuation and Transport Modeling for Application to High Speed Reacting Flows," AIAA Paper No. AIAA-2006-1452, 44th Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 9-12, 2006.

[27] Mattick, S., Brinckman, K.W., and Dash, S.M., "Analysis of High Speed Fuel/Air Mixing Experiments with Improved Scalar Fluctuation Model," 46th Aerospace Sciences Meeting and Exhibit, AIAA Paper No. AIAA-2006-0768, Reno, NV, 7-10 Jan. 2008.

[28] Brinckman, K.W., and Dash, S.M., "Advances in Scalar Fluctuation Modeling and Automated Validation Tools for High-Speed Propulsion Applications," 15th AIAA International Space Planes and Hypersonic Systems and Technologies Conference, AIAA Paper No. AIAA-2008-2534, Dayton, OH, 28th April – 1 May, 2008.