16th Australasian Fluid Mechanics Conference
Crown Plaza, Gold Coast, Australia
2-7 December 2007

Simulation of a sonic jet injected into a supersonic cross-flow

K. Higgins and S. Schmidt

Air Vehicles Division
Defence Science and Technology Organisation, 506 Lorimer Street, Fishermans Bend, Victoria, 3207 AUSTRALIA

Abstract

A hybrid RANS–LES approach is used to simulate the evolution of a sonic underexpanded transverse 4 mm diameter round air jet injected into a Mach 1.6 air cross-flow. Important features of the flow, including a bow shock wave, barrel shock, Mach disk and large-scale unsteady vortical structures in the jet-free-stream shear layer, are similar to those observed in previous experimental studies. A small recirculation region emerges upstream of the jet owing to separation of the approaching boundary layer. This generates a 'necklace' vortex that wraps around the jet and later interacts with the stream-wise-orientated counter-rotating vortex pair within the jet plume. Contours of Reynolds stresses and turbulent kinetic energy from the simulation were compared with experimental measurements. Reasonable qualitative agreement was observed, but the simulation tended to under-predict the peak values. Therefore, the velocity fluctuations recorded in the simulation are somewhat smaller than those measured experimentally. It is likely that this reduced unsteadiness is caused by a lack of grid resolution.

Introduction

The efficient mixing and reaction of fuel and air inside supersonic combustion chambers is critical for the success of hypersonic airbreathing propulsion systems. These mixing processes must be rapid owing to the very short residence times within such combustors. One method to enhance mixing is the sonic injection of fuel from an injector port. Flow visualizations of transverse sonic underexpanded jets in cross-flows obtained by VanLerberghe *et al.*[7] and Ben-Yakar *et al.*[2] show that they are dominated by intermittent large-scale coherent structures in the jet shear layer. Ben-Yakar *et al.*[2] found that the evolution of these structures greatly affected the transverse penetration and mixing of the jet. Therefore, optimising the performance of this method of fuel injection requires detailed knowledge of the turbulent mixing processes that occur when a sonic underexpanded jet interacts with a supersonic cross-flow.

The aim of this paper is to use a hybrid Reynolds-averaged Navier–Stokes (RANS) and large-eddy simulation (LES) approach [5, 6] to investigate the properties of a sonic underexpanded transverse 4 mm diameter round air jet injected into a Mach 1.6 air cross-flow. This configuration, shown schematically in figure 1, was studied experimentally by Santiago [4]. In this experiment, the jet obstructs the cross-flow and produces a three-dimensional bow shock. A small recirculation region emerges upstream of the jet owing to separation of the approaching wall boundary layer, while another separation region emerges immediately downstream of the jet. After leaving the orifice, the jet expands through a Prandtl-Meyer fan before it is compressed through a barrel shock and a Mach disk. The jet is then rapidly turned downstream, and it becomes dominated by a stream-wise-orientated counter-rotating vortex pair. The jet plume cross-section also grows owing to mixing with the cross-flow.

In hybrid RANS–LES methods, large-scale unsteady structures are captured by the LES, while RANS is used in the wall regions. This approach substantially reduces near-wall resolution requirements. The present hybrid RANS–LES investigation was motivated by the work of Peterson *et al.*[3], who used a hybrid RANS–LES unstructured finite volume solver to perform a simulation of the Santiago [4] experiment. The Peterson *et al.*[3] simulation was more sophisticated than the one attempted here owing to the use of a synthetic inflow boundary layer containing unsteady hairpin-vortex structures. Nevertheless, even without the synthetic inflow, they observed significant unsteadiness in the computed jet plume. Although it repeats the Peterson *et al.*[3] investigation, the present study assesses the capability of two different codes to simulate injection into cross-flows. The first of these is a finite-volume structured mesh Navier–Stokes code based on the method of White and Morrison [8]. Various hybrid RANS–LES models can be invoked in the code, some of which are discussed by Baurle *et al.*[1]. The second is the commercial CFD code FLUENT.

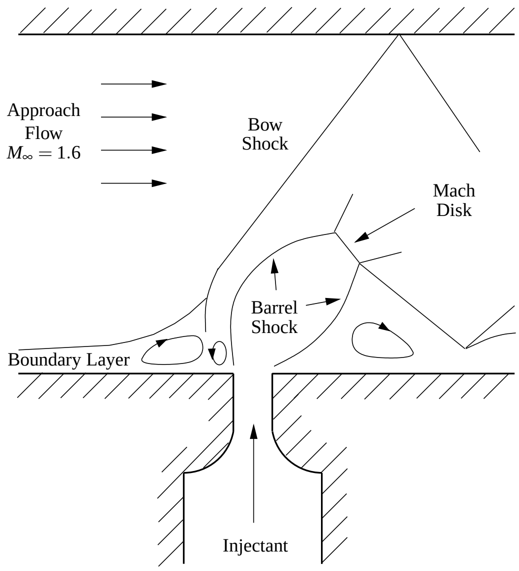

Figure 1: Schematic of injection into a supersonic cross-flow.

Hybrid RANS–LES trials

Following Peterson *et al.*[3], the computational domain for the present simulations replicated the geometry of the Santiago [4] experiment. Here an injector with diameter $d = 4$ mm was located on the centreline of the lower surface of a wind tunnel duct of width 19.05d and height 8.25d. The computational domain extended 5d upstream and 7d downstream of the injector, and the injector plenum chamber and converging nozzle were also included. The computational domain was divided into a structured mesh of 57 blocks containing 2.9×10^6 cells. Symmetry boundary conditions were applied on the side walls of the domain, and adiabatic no-slip boundary conditions with wall functions were applied on the upper and lower walls and in

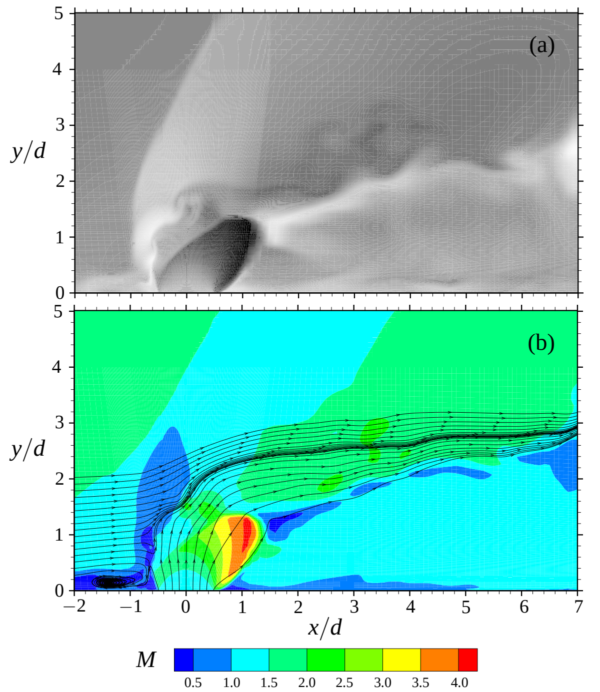

Figure 2: (a) Instantaneous static temperature and (b) Mach number and streamlines from the FLUENT hybrid RANS–LES on the stream-wise centreline plane.

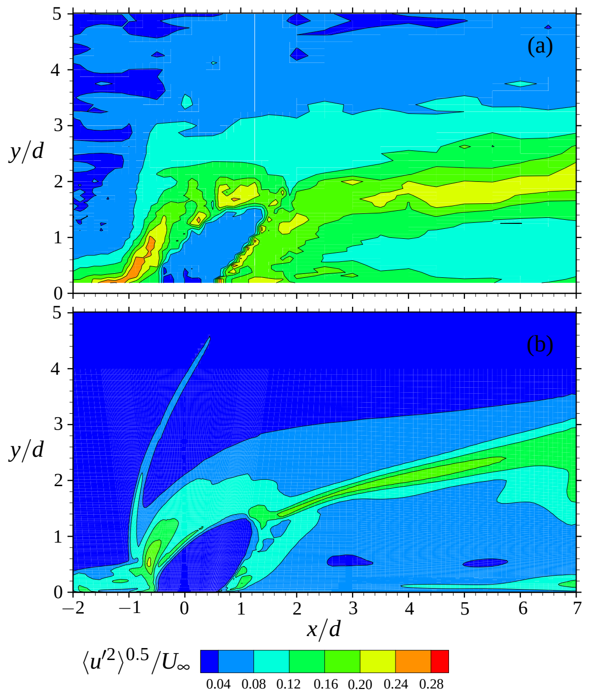

Figure 3: Dimensionless normal Reynolds stress $\langle u'^2\rangle^{0.5}/U_\infty$ on the stream-wise centreline plane. (a) Santiago [4] experiment. (b) FLUENT hybrid RANS–LES.

the plenum chamber and injector nozzle. In order to match the experimental conditions, the inflow boundary condition with a 3.1 mm thickness boundary layer was obtained from a separate steady RANS calculation. The free-stream stagnation conditions for this calculation are summarized in table 1. Stagnation properties used in the boundary condition applied to the base of the plenum chamber are also listed. Supersonic outflow boundary conditions were applied to the downstream face of the computational domain. Table 1 also lists the two important time scales associated with the jet in cross-flow, where the free-stream velocity was taken as $U_\infty \approx 440\,\mathrm{m\,s^{-1}}$. Finally, the computational domain was initialized by computing a steady RANS solution of the jet in cross-flow. The hybrid RANS–LES was then started from this steady solution.

Free-stream Mach number	M_∞	1.6
Free-stream temperature	$T_{0,\infty}$	295 K
Free-stream pressure	$p_{0,\infty}$	241 kPa
Plenum temperature	$T_{0,j}$	300 K
Plenum pressure	$p_{0,j}$	476 kPa
Plenum density	$\rho_{0,j}$	$5.528\,\mathrm{kg\,m^{-3}}$
Cross-flow residence time	$t_r = 12d/U_\infty$	$1.1\times10^{-4}\,s$
'Jet' time scale	$t_j = d/U_\infty$	$9.0\times10^{-6}\,s$

Table 1: Simulation properties.

The first hybrid RANS–LES trial of the jet in cross-flow was calculated using the finite-volume structured mesh Navier–Stokes code. This trial used the well-known Menter-SST turbulence model and a two-equation variant of the hybrid RANS–LES model of Strelets[6] with a DES constant of 0.61. A second-order time accurate diagonalized approximate factorization dual time-stepping scheme was used with a time step of 2.5×10^{-7} s. At each time step, 12 sub-iterations were performed in an attempt to reduce the residual error by about two orders of magnitude. For the FLUENT hybrid RANS–LES trial, the realizable k–ε turbulence model was used in conjunction with a two-equation variant of the hybrid RANS–LES model of Strelets[6] with a DES constant of 0.61. A second-order accurate time stepping scheme was used with a time step of 5×10^{-7} s, and 20 sub-iterations were performed at each time step in an attempt to reduce the residual error by three orders of magnitude.

Results

The first hybrid RANS–LES trial using the finite-volume structured mesh Navier–Stokes code was only partially successful. The structure of the flow was found to be well represented, including the upstream recirculation region, bow and barrel shocks and Mach disk. However, a problem was identified after examining a time sequence of static temperature contours on the stream-wise centreline plane. These showed little unsteadiness over a period of 3.2 cross-flow residence times. The time-averaged Reynolds stresses obtained from the trial showed that the velocity fluctuations were confined to the jet-free-stream shear layer downstream of the Mach disk. No unsteadiness was observed in the recirculation region upstream of the injector. These results were at variance with the experimental measurements of Santiago [4]. It was thought that the observed damping of the natural unsteadiness of the jet in cross-flow might be caused by numerical dissipation in the code, or perhaps by a lack of grid resolution near the injector.

The second hybrid RANS–LES trial using FLUENT was somewhat more successful. In this case, the simulation was advanced by 200 time steps. This is equivalent to 0.9 cross-flow residence times. Figure 2(a) shows the static temperature field on the stream-wise centreline plane from a representative instant during the simulation. Turbulent structures can be seen emerging between the bow and barrel shocks, and these interact with the jet-free-stream shear layer further downstream. These observations are in good agreement with flow visualizations ob-

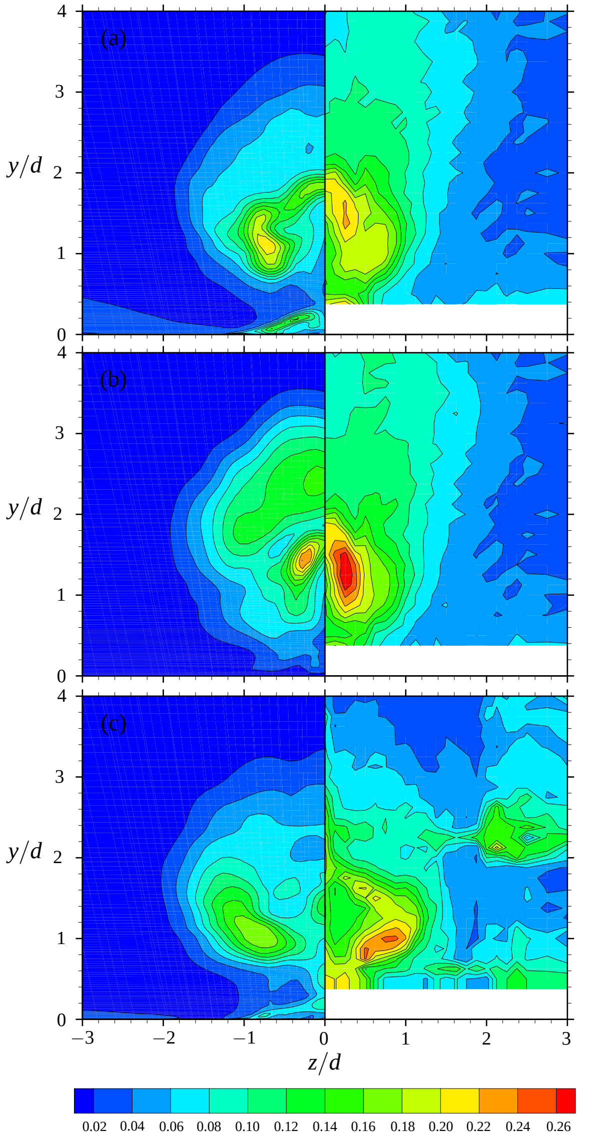

Figure 4: Dimensionless normal Reynolds stresses on the span-wise plane $x/d = 3$: FLUENT hybrid RANS–LES (left) and Santiago [4] experiment (right). (a) $\langle u'^2\rangle^{0.5}/U_\infty$, (b) $\langle v'^2\rangle^{0.5}/U_\infty$, (c) $\langle w'^2\rangle^{0.5}/U_\infty$.

tained by VanLerberghe [7]. Contours of the Mach number overlaid with streamlines on the stream-wise centreline plane are shown in figure 2(b). Here the jet emerges at sonic velocity and expands to at least Mach 4 before passing through the Mach disk. Jet fluid also moves through the upper surface of the barrel shock, and a recirculation region emerges upstream of the injector.

Statistics were collected from the FLUENT hybrid RANS–LES trial over the last 100 time steps, or 5.6 'jet' time scales. This period was much shorter than the 87 'jet' time scales over which Peterson *et al.*[3] collected statistics. Although the present statistics can be compared with Santiago's [4] measurements, more data needs to be collected. This is a work in progress. Figure 3 compares the dimensionless normal stream-wise Reynolds stress on the stream-wise centreline plane from Santiago's [4] experiment with the present hybrid RANS–LES result. Overall, the simulation shows a significantly lower level of fluctuation in the stream-wise velocity component compared with the experiment. The peak $\langle u'^2\rangle/U_\infty$ observed falls 26% below the peak experimental measurement. Velocity fluctuations are noticeably smaller in the vicinity of the upstream recirculation region, and this attenuates an important source of instability-inducing perturbations. Consequently, unsteadiness in the jet-free-stream shear layer downstream of the barrel shock is less intense and confined to a narrower region compared with the experimental result.

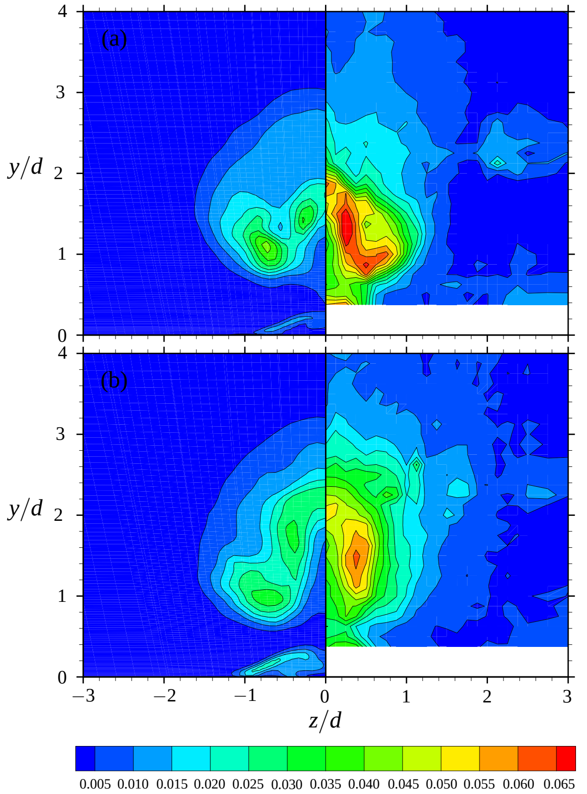

Figure 5: Dimensionless TKE on the span-wise planes (a) $x/d = 3$ and (b) $x/d = 5$ from the FLUENT hybrid RANS–LES (left) and Santiago [4] experiment (right).

Figure 4 shows dimensionless normal Reynolds stresses on the span-wise plane $x/d = 3$ from the present hybrid RANS–LES compared with the Santiago [4] experimental measurements. Reasonable qualitative agreement is observed, but the peak Reynolds stresses and their spatial extent are somewhat under-predicted by the present simulation. The 'horse shoe' shape of the contours is caused by the presence of a stream-wise orientated counter-rotating vortex pair in the jet plume. Visualizations (not included here) showed that these vortex structures also interacted with the 'necklace' vortex (generated by separation of the boundary layer upstream of the injector) that wrapped around the windward side of the jet and trailed downstream. Figure 5 shows that similar qualitative agreement and quantitative under-prediction are obtained from a comparison of contours of dimensionless turbulent kinetic energy (TKE) $(\langle u'^2\rangle + \langle v'^2\rangle + \langle w'^2\rangle)/(2U_\infty)$ with experimental results on the span-wise planes with $x/d = 3$ and $x/d = 5$.

RANS predicts mean flow quantities using turbulence models that model the entire turbulent spectrum. These models do not account for the large, three-dimensional eddying motions captured by LES. Therefore, a key advantage of a hybrid RANS–LES approach over RANS is the possibility that mean quantities like velocity or temperature can be more accurately pre-

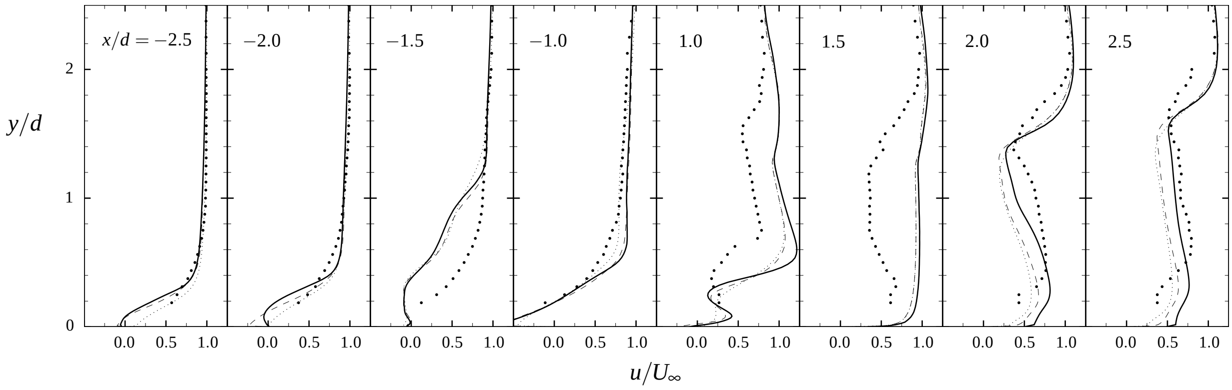

Figure 6: Dimensionless time-averaged stream-wise velocity profiles at various x/d stations on the stream-wise centreline plane: ——— , FLUENT hybrid RANS-LES (rke); ········ , RANS (rke); – – – – , RANS (SST k−ω); • , Santiago[4] experiment.

dicted in flows containing large eddying motions. However, given that the present hybrid RANS–LES tends to under-predict velocity fluctuations, this advantage could be lost. One way to investigate this point is to compare mean velocity profiles from the hybrid RANS–LES with those from steady RANS. Figure 6 shows such a comparison of dimensionless mean velocity profiles at various stations on the stream-wise centreline plane. Here RANS results using two different turbulence models (realizable $k-\varepsilon$ and SST $k-\omega$) are shown. Experimental mean velocity profiles measured by Santiago [4] are also shown. Overall, the hybrid RANS–LES profiles are very similar to the RANS results, and improved agreement with the experimental data is not obtained.

There are several reasons why improved agreement might not be achieved. First, the inflow boundary layer does not match the experimental profile exactly. Secondly, from the profiles at $x/d = -2.0$ and -1.5, it would appear that separation tends to occur earlier in both the RANS and hybrid RANS–LES compared with the experiment. The similarity of the RANS and hybrid RANS–LES profiles here also suggests that RANS dominates in the recirculation region, giving a plausible reason why separation is inadequately modelled. Grid refinement upstream of the injector would increase the use of LES within the recirculation region, and might allow the growth of natural instabilities that would otherwise be suppressed by RANS alone. This is likely to yield a more accurate model of separation, and also a source of perturbations that would increase the level of unsteadiness observed downstream in the jet plume. Improved agreement between mean velocity profiles obtained from hybrid RANS–LES and experiment would then be expected. A grid refinement study is presently underway.

Conclusions

A hybrid RANS–LES approach was used to simulate the evolution of a sonic underexpanded transverse 4 mm diameter round air jet injected into a Mach 1.6 air cross-flow. In the simulation, large-scale unsteady structures were captured by the LES, while RANS was used in the wall regions. This approach substantially reduces near-wall resolution requirements. However, coarse grid resolution near the injector may suppress the natural unsteadiness of the jet. The present simulation showed that the jet obstructed the cross-flow, producing a three-dimensional bow shock-wave. A small recirculation region emerged upstream of the jet owing to separation of the approaching boundary layer. After leaving the orifice, the jet expanded through a Prandtl-Meyer fan before it was compressed through a barrel shock and a Mach disk. The jet was then rapidly turned downstream, becoming dominated by a stream-wise-orientated counter-rotating vortex pair. Contours of static temperature on the stream-wise centreline plane showed the formation of large-scale unsteady vortical structures in the jet-free-stream shear layer. These observations were in good agreement with flow visualization obtained by VanLerberghe [7] of a sonic jet in a Mach 1.6 cross-flow that had been studied experimentally by Santiago [4].

Contours of Reynolds stresses and turbulent kinetic energy from the simulation were compared with experimental measurements made by Santiago [4]. Reasonable qualitative agreement was observed, but the simulation tended to under-predict the peak values. Additionally, the unsteadiness in the jet-free-stream shear layer downstream of the barrel shock in the simulation was confined to a narrower region compared with the experimental result. Contours of Reynolds stresses and turbulent kinetic energy on span-wise planes at distances $3d$ and $5d$ downstream of the injector (where d is its diameter) revealed ‘horse shoe’-shaped structures related to unsteadiness in the stream-wise-orientated counter-rotating vortex pair. The peak values observed and their spatial extent all somewhat under-estimated the experimental results. A comparison of mean velocity profiles also showed that the hybrid RANS–LES results were similar to RANS, and that improved agreement with experimental data was not obtained. It is possible that the reduced unsteadiness observed in the hybrid RANS–LES, compared with the experiment, was caused by a lack of computational grid resolution. A grid refinement study is presently underway with the aim of improving the prediction of separation upstream of the injector. It is hoped that this will provide a stronger source of upstream perturbations to increase unsteadiness within the downstream jet plume.

Acknowledgements

We would like to thank Professor Juan Santiago from Stanford University for providing the experimental data, and David Peterson from University of Minnesota for helpful advice.

References

[1] Baurle, R. A., Tam, C. J., Edwards, J. R. and Hanson, H. A., Hybrid simulation approach for cavity flows: blending, algorithm and boundary treatment issues, *AIAA Journal*, **41**, 2003, 1463–1480.

[2] Ben-Yakar, A., Mungal, M. G. and Hanson, R. K., Time evolution and mixing characteristics of hydrogen and ethy-

lene transverse jets in supersonic crossflows, *Phys. Fluids*, **18**, 2006, 026101.

[3] Peterson, D. M., Subbareddy, P. K. and Candler, G. V., Assessment of synthetic inflow generation for simulating injection into a supersonic crossflow, *AIAA 2006-8128*.

[4] Santiago, J. C., *An experimental study of the velocity field of a transverse jet injected into a supersonic crossflow*, Ph.D. thesis, University of Illinois, Urbana-Champaign, 1995.

[5] Spalart, P. R., Jou, W. H., Strelets, M. and Allmaras, S. R., Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach, in *Advances in DNS/LES*, editors C. Liu and Z. Liu, Greyden, Columbus, 1997.

[6] Strelets, M., Detached eddy simulation of massively separated flows, *AIAA 2001-0879*.

[7] VanLerberghe, W. M., Santiago, J. G., Dutton, J. C. and Lucht, R. P., Mixing of a sonic transverse jet injected into a supersonic flow, *AIAA Journal*, **38**, 2000, 470–479.

[8] White, J. A. and Morrison, J. H., A pseudo-temporal multigrid relaxation scheme for solving the parabolized Navier–Stokes equations, *AIAA 99-3360*.