where there is a pressure peak as seen in the theoretical model of the roller-type structure and recent experimental measurements.[12] This supports the idea that the excitation causes a pressure peak in the mixing layer that induces the roller-type structure.

Instantaneous and average images were also taken for a Mach 2 jet ($M_c = 0.85$) with no excitation (Figs. 4a–4c) and with laser excitation (Figs. 4d and 4e) having a delay time of 100 μs between the excitation and imaging pulses. For the unforced case, the decrease in mixing layer growth rate is clearly observed compared to the lower-convective-Mach-numbercase shown earlier. Also there is no indication of organized large-scale, roller-type structures. With laser excitation, however, a spatially stable, roller-type structure is observed in the instantaneous and average images with distinct core ($39 \pm 4\%$ thicker than the unforced case) and braid regions, as observed for the lower-Mach-number case. The convective velocity measured for this case is 365 m/s for the core and 309 m/s for the braid region, with an uncertainty of ± 4 m/s. Again the measured convective velocity from the braid region is closer to the theoretical convective velocity of 297 ± 5 m/s, with the measured value slightly higher as already discussed.

The forcing mechanism for the large-scale structures in the present study may be similar to wall heating used to force Tollmien–Schlichting waves in subsonic boundary layers[13] and subsequent Kelvin–Helmholtz waves in free shear layers.[14] Liepmann et al.[13] reported that the forcing of the instability can be explained by the boundary-layer momentum equation and by observing that heating the surface (for a gas) has the same effect as an adverse pressure gradient. If the temperature is large enough, this could lead to a local region of separation, which may force the large-scale structures observed in the present experiments. Note, however, that the forcing was not large enough to cause the nozzle to unstart or create strong shocks, which would have appeared in instantaneous schlieren images that were taken. Further experiments are needed, however, to confirm this explanation.

IV. Conclusion

An innovative method of controlling and forcing the creation of large roller-type structures in compressible mixing layers is presented. A laser beam from a pulsed Nd:YAG laser is focused on the nozzle exit of axisymmetric supersonic jets with Mach numbers of 1.36 and 2, resulting in convective Mach numbers of 0.63 and 0.85, respectively. Laser excitation causes a thermal bump at the wall, which induces the formation of a roller-type structure in the mixing layer. Instantaneous and phase-averaged images were taken of the mixing layer to investigate the ability to induce the large-scale structure and measure its characteristics. The convective velocities of the core and braid regions of the structure were found to be higher than the theoretical values. The thickness of the core region was found to be from 32 to 38% greater than the shear layer thickness without excitation. Future experiments will be conducted to verify the mechanism inducing the formation of the roller-type structure and to investigate other excitation beam geometries and multiple pulse forcing.

Acknowledgments

The authors acknowledge the support of U.S. Air Force Office of Scientific Research Grant 96-0144 with Mark Gruber, National Science Foundation Grant NSF-9622108 with R. Arndt, and a Rutgers Research Council Grant.

References

[1]Papamoschou, D., and Roshko, A., "The Compressible Turbulent Mixing Layer: An Experimental Study," *Journal of Fluid Mechanics*, Vol. 197, Dec. 1988, pp. 453–477.

[2]Elliott, G. S., and Samimy, M., "Compressibility Effects in Free Shear Layers," *Physics of Fluids A*, Vol. 2, No. 7, 1990, pp. 1231–1240.

[3]Goebel, S. G., and Dutton, J. C., "Experimental Study of Compressible Turbulent Mixing Layers," *AIAA Journal*, Vol. 29, No. 4, 1991, pp. 538–545.

[4]Clemens, N. T., and Mungal, M. G., "Two- and Three-Dimensional Effects in the Supersonic Mixing Layer," *AIAA Journal*, Vol. 30, No. 4, 1992, pp. 973–981.

[5]Papamoschou, D., "Structure of the Compressible Turbulent Shear Layer," *AIAA Journal*, Vol. 29, No. 5, 1991, pp. 680, 681.

[6]Hall, J. L., Dimotakis, P. E., and Rosemann, H., "Experiments in Nonreacting Compressible Shear Layers," *AIAA Journal*, Vol. 31, No. 12, 1993, pp. 2247–2254.

[7]Gutmark, E. J., Schadow, K. C., and Yu, K. H., "Mixing Enhancement in Supersonic Free Shear Flows," *Annual Review of Fluid Mechanics*, Vol. 27, 1995, pp. 375–417.

[8]Elliott, G. S., Samimy, M., and Arnette, S. A., "Study of Compressible Mixing Layers Using Filtered Rayleigh Scattering Based Visualizations," *AIAA Journal*, Vol. 30, No. 10, 1992, pp. 2567–2569.

[9]Schmisseur, J. D., Schneider, S. P., and Collicott, S. H., "Receptivity of the Mach-4 Boundary-Layer on an Elliptic Cone to Laser-Generated Localized Freestream Perturbations," AIAA Paper 98-0532, Jan. 1998.

[10]Tsao, J. Y., Picraux, S. T., Peercy, P. S., and Thompson, M. O., "Direct Measurements of Liquid/Solid Interface Kinetics During Pulsed-Laser-Induced Melting of Aluminum," *Applied Physics Letters*, Vol. 48, No. 4, 1986, pp. 278–280.

[11]Dimotakis, P. E., "On the Convection Velocity of Turbulent Structures in Supersonic Shear Layers," AIAA Paper 91-1724, June 1991.

[12]Elliott, G. S., Crawford, J. B., and Mosedale, A. D., "Enhancement of Large Scale Structures in Supersonic Axisymmetric Jets Using Laser Excitation," AIAA Paper 98-0532, Jan. 1998.

[13]Liepmann, H. W., Brown, G. L. and Nosenchuck, D. M., "Control of Laminar-Instability Waves Using a New Technique," *Journal of Fluid Mechanics*, Vol. 118, May 1982, pp. 187–200.

[14]Gharib, M., "Response of the Cavity Shear Layer Oscillations to External Forcing," *AIAA Journal*, Vol. 25, No. 1, 1987, pp. 43–47.

F. W. Chambers
Associate Editor

Artificial Dissipation Schemes for Viscous Airfoil Computations

K. Frew,* D. W. Zingg,† and S. De Rango*
University of Toronto,
Downsview, Ontario M3H 5T6, Canada

Introduction

IT is generally accepted that, in solving the Navier–Stokes equations for practical high-Reynolds-number turbulent flows, the spatial discretization must contain some sort of numerical dissipation, either implicitly, as in upwind schemes, or explicitly, through an artificial dissipation scheme. The dissipation is required to prevent high-frequency oscillations, especially when the flowfield contains shock waves, and to provide stability. Dissipation is needed even in viscous flows because they are essentially underresolved, that is, the shortest wavelengths present in the real flow, which are limited by viscosity, are generally not resolved using practical meshes.

Several researchers have shown that the popular scalar artificial dissipation scheme[1] can be a major source of numerical error, especially in laminar boundary layers and in drag prediction.[2–4] This problem is reduced with upwind schemes and matrix dissipation.[5] However, the scalar dissipation scheme is inexpensive and easy to implement. Consequently, some researchers have proposed scalings for the scalar dissipation scheme in an attempt to reduce numerical errors. These include scalings based on local Mach number and vorticity.[6]

The purpose of this Note is to examine the effect of the artificial dissipation scheme on the prediction of lift and drag in thin-layer Navier–Stokes computations of subsonic and transonic airfoil flows. In addition to the matrix dissipation scheme of Swanson and Turkel,[5] we present and evaluate a new scaling of the scalar

Received June 10, 1996; presented as Paper 96-1970 at the AIAA 27th Fluid Dynamics Conference, New Orleans, LA, June 17–20, 1996; revision received May 18, 1998; accepted for publication May 19, 1998.

*Graduate Student, Institute for Aerospace Studies, 4925 Dufferin Street.

†Associate Professor, Institute for Aerospace Studies, 4925 Dufferin Street. Member AIAA.

dissipation scheme, which is based on the cell Reynolds number. Further details of the present study can be found in Ref. 7.

The artificial dissipation schemes have been implemented in the thin-layer Navier–Stokes solver ARC2D.[8] This solver uses second-order centered differences in space through a generalized curvilinear coordinate transformation and is, thus, applicable to structured grids. Convergence to steady state is achieved using the diagonal form of the Beam and Warming approximate-factorization algorithm with local time stepping. The Baldwin–Lomax turbulence model is used with C_{wk} set to 1.0 (rather than 0.25).

Matrix Dissipation Scheme

The matrix dissipation scheme is typically presented through the connection with upwind schemes. Here we develop the idea in a slightly different manner. Any difference operator, including one-sided and biased operators, can be written as the sum of an antisymmetric component δ_a and a symmetric component δ_s. The antisymmetric component corresponds to a centered difference, whereas the symmetric component provides dissipation. Consider the linear convection equation given by

$$\frac{\partial u}{\partial t} + c\frac{\partial u}{\partial x} = 0 \tag{1}$$

where c is a real scalar constant. To provide dissipation, the spatial operator must be applied in the following manner:

$$c\frac{\partial u}{\partial x} \approx c\delta_a u + |c|\delta_s u \tag{2}$$

To extend this to a linear constant-coefficient hyperbolic system in the form

$$\frac{\partial q}{\partial t} + A\frac{\partial q}{\partial x} = 0 \tag{3}$$

where A is the flux Jacobian matrix, we first diagonalize the system, giving

$$\frac{\partial w}{\partial t} + \Lambda\frac{\partial w}{\partial x} = 0 \tag{4}$$

where $w = X^{-1}q$, $\Lambda = X^{-1}AX$, and X is the matrix of right eigenvectors. From this it is clear that the spatial operator should be applied as follows:

$$\Lambda\frac{\partial w}{\partial x} \approx \Lambda\delta_a w + |\Lambda|\delta_s w \tag{5}$$

based on Eq. (2). Premultiplying by X and replacing w with $X^{-1}q$ gives

$$A\frac{\partial q}{\partial x} \approx A\delta_a q + |A|\delta_s q \tag{6}$$

where $|A| = X|\Lambda|X^{-1}$. Thus, the symmetric component of the operator is multiplied by the matrix $|A|$; hence, the designation matrix dissipation. The scalar dissipation scheme is obtained by approximating $|A|$ by the spectral radius of A, thus reducing the computational expense of the scheme.

Details of the present implementation of the matrix dissipation scheme, which is based on the work of Swanson and Turkel,[5] are given in Ref. 7. Near stagnation points and sonic points, some of the eigenvalues of the flux Jacobians approach zero. To avoid the problems this may introduce, the eigenvalues are modified as follows:

$$\tilde{\lambda}_1, \tilde{\lambda}_2 = \max(\lambda_{1,2}, V_l\sigma), \qquad \tilde{\lambda}_3 = \max(\lambda_3, V_n\sigma)$$
$$\tilde{\lambda}_4 = \max(\lambda_4, V_n\sigma) \tag{7}$$

Here σ is the spectral radius of the flux Jacobian, $\lambda_{1,2}$ are the convective eigenvalues, and $\lambda_{3,4}$ are the eigenvalues containing the sound speed. The values of V_l and V_n used are discussed further subsequently. When $V_l = V_n = 1.0$, the scalar artificial dissipation scheme is obtained.

Cell Reynolds Number Scaling

The cell Reynolds number Re_Δ provides a measure of the degree to which viscous effects are resolved. In regions where the cell Reynolds number is small, viscous effects are well resolved and, thus, the need for artificial dissipation is reduced. Hence, the cell Reynolds number provides a suitable parameter to scale the scalar artificial dissipation coefficient. In contrast to other scaling parameters, such as vorticity, which sense regions where viscous effects are significant, the cell Reynolds number incorporates the grid resolution as well.

The present scaling is given as a smooth function of the cell Reynolds number between an upper and lower limit. For cell Reynolds numbers above the upper limit, the full scalar coefficient is used. Below the lower limit, the artificial dissipation is switched off. We have used a lower limit r_l equal to 10 and an upper limit r_u equal to 1000. No scaling is applied in the streamwise ξ direction. The following scaling is used in the normal η direction:

$$\phi_{j,k} = \begin{cases} 0 & Re_\Delta \le r_l \\ 1 & Re_\Delta \ge r_u \\ f(Re_\Delta) & r_l \le Re_\Delta \le r_u \end{cases} \tag{8}$$

with

$$Re_\Delta = \frac{\rho\sqrt{u^2+v^2}\,\delta n}{\mu + \mu_t}Re_\infty, \qquad \delta n = J^{-1}\sqrt{\xi_x^2 + \xi_y^2} \tag{9}$$

where Re_∞ is the freestream Reynolds number, which is introduced because the physical quantities have been nondimensionalized, and μ_t is the turbulent eddy viscosity. The Jacobian J^{-1} and the metrics ξ_x and ξ_y arise from the transformation to generalized curvilinear coordinates. The function $f(Re_\Delta)$ is given by

$$f(Re_\Delta) = a_1 + a_2 Re_\Delta + a_3 Re_\Delta^2 + a_4 Re_\Delta^3$$
$$a_1 = \frac{r_l^3 - 3r_u r_l^2}{a_5}, \qquad a_2 = \frac{6r_l r_u}{a_5}, \qquad a_3 = -\frac{3(r_l + r_u)}{a_5} \tag{10}$$
$$a_4 = \frac{2}{a_5}, \qquad a_5 = r_l^3 - 3r_l^2 r_u + 3r_l r_u^2 - r_u^3$$

Results and Discussion

To compare the artificial dissipation options, three flows over the NACA 0012 airfoil are examined using three different grids. The conditions for the flow cases are as follows: 1) $M_\infty = 0.16$, $\alpha = 0$ deg, $Re = 2.88 \times 10^6$, transition at 0.43 chords on both surfaces; 2) $M_\infty = 0.16$, $\alpha = 6$ deg, $Re = 2.88 \times 10^6$, transition at 0.05 and 0.8 chords on the upper and lower surfaces, respectively; and 3) $M_\infty = 0.7$, $\alpha = 3.0$ deg, $Re = 9 \times 10^6$, transition at 0.05 chords on both surfaces.

All of the grids have a C topology. The grid designated A7B has 249 points in the streamwise direction with 97 points in the normal direction for a total of roughly 25,000 grid points. There are 201 grid points on the airfoil surface and 25 points along the wake cut. Points are clustered in the streamwise direction at the leading and trailing edges with a spacing of 0.0002 chords. The normal spacing to the first grid line off of the surface is 2×10^{-7} chords. The distance to the outer boundary is 12 chords. The grid designated N7B is a 249×49 grid, i.e., roughly 12,500 nodes, with the same streamwise point distribution and off-wall spacing as grid A7B. Finally, the grid designated N6B has the same parameters as grid N7B except that the off-wall spacing is roughly 2×10^{-6} chords.

These flows were studied in Refs. 9 and 10. Based on the data presented in the latter, in which grid studies were performed using grids with over 200,000 nodes, we obtain estimates of the grid-independent values of the drag components and the lift (for the present turbulence model and outer boundary location). These

Table 1 Grid study for case 1[a]

Grid	Dissipation	$\%C_d$	$\%C_{d_p}$	$\%C_{d_f}$
A7B	Matrix	0.0	0.0	0.0
A7B	$Re_\triangle$	0.1	2.6	−0.3
A7B	Scalar	3.6	6.3	3.0
N7B	Matrix	−3.9	3.2	−5.1
N7B	$Re_\triangle$	−5.1	18.2	−8.8
N7B	Scalar	14.2	41.4	10.0
N6B	Matrix	−2.5	0.6	−2.9
N6B	$Re_\triangle$	−3.4	11.6	−5.8
N6B	Scalar	42.0	41.3	43.9

[a]Grid-independent $C_d = 0.00579$, $C_{d_p} = 0.00080$, and $C_{d_f} = 0.00499$.

Table 2 Grid study for case 2[a]

Grid	Dissipation	$\%C_l$	$\%C_d$	$\%C_{d_p}$	$\%C_{d_f}$
A7B	Matrix	0.4	5.6	17.2	−0.2
A7B	$Re_\triangle$	0.5	5.1	16.6	−0.6
A7B	Scalar	−0.3	8.8	18.8	3.2
N7B	Matrix	−1.2	16.9	62.1	−5.4
N7B	$Re_\triangle$	−1.3	15.8	63.9	−7.9
N7B	Scalar	−0.8	30.1	75.4	7.4
N6B	Matrix	−0.9	13.9	47.8	−2.8
N6B	$Re_\triangle$	−0.6	12.8	48.2	−4.6
N6B	Scalar	−1.2	47.4	59.2	41.6

[a]Grid-independent $C_l = 0.6620$, $C_d = 0.00787$, $C_{d_p} = 0.00260$, and $C_{d_f} = 0.00527$.

values, which are given in Tables 1–3, are used to estimate numerical errors.

As shown in Ref. 7, the different artificial dissipation schemes have little effect on the convergence rate. However, the matrix scheme leads to a 15–20% increase in computing time per iteration, whereas the $Re_\triangle$ scaling requires only a 1% increase compared with the scalar scheme. To be cost effective, the matrix scheme must require significantly fewer grid nodes to achieve a given level of accuracy.

Table 1 shows the numerical errors in drag obtained for case 1. The entries show the percent difference in the drag coefficient C_d, the pressure drag component C_{d_p}, and the friction drag component C_{d_f} from the estimated grid-independent result given in the footnote. The matrix dissipation results shown were computed with $V_n = V_l = 0$. Both matrix dissipation and $Re_\triangle$ scaling lead to a marked reduction in error. This is especially evident on the coarser grids, N7B and N6B, on which the scalar dissipation scheme produces very large errors. The use of an off-wall spacing on the order of 2×10^{-7} (which produces a y^+ value much less than unity) is not common. However, the error in friction drag produced with the scalar dissipation scheme is substantially reduced using the smaller off-wall spacing. For the matrix scheme and the $Re_\triangle$ scaling, the errors obtained on grid N6B are smaller than those on grid N7B. On the basis of this case, for which friction drag is the dominant component, one would definitely conclude that the matrix scheme is cost effective relative to the scalar scheme and that the $Re_\triangle$ scaling is a promising option.

Table 2 shows the estimates of the numerical errors in lift, drag, and the individual drag components for case 2. Numerical errors in lift are small. Errors in drag are much larger. Matrix dissipation and $Re_\triangle$ scaling consistently produce a reduction in friction drag error with a smaller reduction in pressure drag error. This is especially striking on grid N6B. However, the reduction in total drag error is not nearly as great as in case 1, primarily because the reduction in pressure drag error is small. Comparing the matrix scheme's result on grid N6B with that of the scalar scheme on grid N7B, the drag error is reduced by about 50%. On grid A7B, the reduction is about 35%. On grid A7B, it appears that the second-order truncation error is more important than the artificial dissipation for this case.

Errors for case 3 are tabulated in Table 3. For transonic cases, the convergence history can be quite dependent on the values of V_n and V_l. The results shown were computed using $V_l = 0.025$ and

Table 3 Grid study for case 3[a]

Grid	Dissipation	$\%C_l$	$\%C_d$	$\%C_{d_p}$	$\%C_{d_f}$
A7B	Matrix	−0.08	1.2	3.0	−2.0
A7B	$Re_\triangle$	−1.0	1.5	3.6	−2.0
A7B	Scalar	−0.8	0.8	2.3	−1.8
N7B	Matrix	−1.4	4.1	9.3	−4.6
N7B	$Re_\triangle$	−1.4	4.0	9.5	−5.4
N7B	Scalar	−2.1	4.5	9.1	−3.4
N6B	Matrix	−1.2	3.4	6.6	−2.0
N6B	$Re_\triangle$	−1.4	5.3	9.8	−2.5
N6B	Scalar	−2.0	11.0	7.0	18.2

[a]Grid-independent $C_l = 0.509$, $C_d = 0.01353$, $C_{d_p} = 0.00853$, and $C_{d_f} = 0.00500$.

$V_n = 0.25$. For this case, the pressure drag exceeds the friction drag, primarily as a result of the shock wave. The improved dissipation schemes provide very little error reduction. It is possible that the error is largely associated with the first-order dissipation added near the shock wave and, thus, the numerical dissipation in boundary layers is not an important factor.

Conclusions

We have compared the scalar and matrix artificial dissipation schemes and a new scaling of the scalar scheme based on the cell Reynolds number in terms of their ability to predict airfoil lift and drag coefficients. Both the matrix dissipation scheme and the cell Reynolds number scaling reduce errors, especially in friction drag and boundary-layer velocity profiles. Although not demonstrated here, the matrix scheme is particularly effective for laminar flows and, hence, should be used if laminar–turbulent transition is to be predicted based on boundary-layer velocity profiles. However, the impact on the global errors is case dependent. For cases in which the pressure drag is the major component, including high-lift cases and cases with strong shocks, the error reduction associated with the matrix dissipation scheme and the cell Reynolds number scaling is fairly small. The possibility of combining the matrix scheme with the cell Reynolds number scaling should be considered.

References

[1]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge–Kutta Time Stepping," AIAA Paper 81-1259, June 1981.

[2]Turkel, E., and Vatsa, V. N., "Effect of Artificial Viscosity on Three-Dimensional Flow Solutions," *AIAA Journal*, Vol. 32, No. 1, 1994, pp. 39–45.

[3]Allmaras, S. R., "Contamination of Laminar Boundary Layers by Artificial Dissipation in Navier–Stokes Solutions," *Numerical Methods for Fluid Dynamics*, edited by M. J. Baines and K. W. Morton, Clarendon, Oxford, England, UK, 1993.

[4]Varma, R. R., and Caughey, D. A., "Estimation of the Integrated Effect of Numerical Dissipation on Navier–Stokes Solutions," *Fourth International Symposium on Computational Fluid Dynamics*, Davis, CA, 1991, pp. 1161–1166.

[5]Swanson, R. C., and Turkel, E., "On Central-Difference and Upwind Schemes," *Journal of Computational Physics*, Vol. 101, No. 2, 1992, pp. 292–306.

[6]Hall, M. G., "On the Reduction of Artificial Viscosity in Viscous Flow Solutions," *Frontiers of Computational Fluid Dynamics—1994*, edited by D. A. Caughey and M. M. Hafez, Wiley, New York, 1994, pp. 303–317.

[7]Frew, K., and Zingg, D. W., "On Artificial Dissipation Models for Viscous Airfoil Computations," AIAA Paper 96-1970, June 1996.

[8]Pulliam, T. H., "Efficient Solution Methods for the Navier–Stokes Equations," Lecture Series Notes, Von Kármán Inst. for Fluid Dynamics, Brussels, Belgium, Jan. 1986.

[9]Zingg, D. W., "Grid Studies for Thin-Layer Navier–Stokes Computations of Airfoil Flowfields," *AIAA Journal*, Vol. 30, No. 10, 1992, pp. 2561–2564.

[10]Nemec, M., and Zingg, D. W., "Aerodynamic Computations Using the Convective Upstream Split Pressure Scheme with Local Preconditioning," AIAA Paper 98-2444, June 1998.

D. S. McRae
Associate Editor