Computational Aeroacoustics by the Space-Time CE/SE Method

Ching Y. Loh
Taitech, Inc., Cleveland, Ohio

February 2001
Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA’s scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA’s institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA’s counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA’s mission.

Specialized services that complement the STI Program Office’s diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:


- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA Access Help Desk at 301-621-0134

- Telephone the NASA Access Help Desk at 301-621-0390

- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076
Computational Aeroacoustics by the Space-Time CE/SE Method

Ching Y. Loh
Taitech, Inc., Cleveland, Ohio

Prepared for the
Second Aeroacoustics Workshop of Germany (SWING)
sponsored by the Deutsches Zentrum für Luft- und Raumfahrt
Braunschweig, Germany, October 6–7, 2000

Prepared under Contract NAS3-97186

National Aeronautics and
Space Administration

Glenn Research Center

February 2001
1 Introduction

In recent years, a new numerical methodology for conservation laws – the Space-Time Conservation Element and Solution Element Method (CE/SE), was developed by Dr. Chang of NASA Glenn Research Center and collaborators [1-3]. In nature, the new method may be categorized as a finite volume method, where the conservation element (CE) is equivalent to a finite control volume (or cell) and the solution element (SE) can be understood as the cell interface. However, due to its rigorous treatment of the fluxes and geometry, it is different from the existing schemes. The CE/SE scheme features:

(1) space and time treated on the same footing, the integral equations of conservation laws are solved for with second order accuracy,
(2) high resolution, low dispersion and low dissipation,
(3) novel, truly multi-dimensional, simple but effective non-reflecting boundary condition,
(4) effortless implementation of computation, no numerical fix or parameter choice is needed,
(5) robust enough to cover a wide spectrum of compressible flow: from weak linear acoustic waves to strong, discontinuous waves (shocks), appropriate for linear and nonlinear aeroacoustics.

Currently, the CE/SE scheme has been developed to such a stage that a 3-D unstructured CE/SE Navier-Stokes solver is already available. However, in the present paper, as a general introduction to the CE/SE method, only the 2-D unstructured Euler CE/SE solver is chosen as a prototype and is sketched in Section 2. Then applications of the CE/SE scheme to linear, nonlinear aeroacoustics and airframe noise are depicted in Sections 3, 4, and 5 respectively to demonstrate its robustness and capability.

2 the 2-D Unstructured CE/SE Scheme

The CE/SE scheme can be used with either structured or unstructured grids. Here, the unstructured CE/SE is used as a prototype for introductory description.

2.1 Conservation Form of the Unsteady Euler Equations

Consider a dimensionless conservation form of the unsteady Euler equations of a perfect gas. Let \( \rho, u, v, \) and \( \gamma \) be the density, streamwise velocity component, transversal velocity component, static pressure, and constant specific heat ratio, respectively. The 2-D Euler equations then can be written in the following vector form:

\[
U_t + F_x + G_y = 0, \tag{1}
\]

where \( x, y, \) and \( t \) are the streamwise and transversal coordinates and time, respectively. The conservative flow variable vector \( U \) and the flux vectors \( F \) and \( G \), are given by:

\[
U = \begin{pmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{pmatrix}, \quad F = \begin{pmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{pmatrix}, \quad G = \begin{pmatrix} G_1 \\ G_2 \\ G_3 \\ G_4 \end{pmatrix},
\]

with

\[
U_1 = \rho, U_2 = \rho u, U_3 = \rho v, U_4 = p/(\gamma - 1) + \rho(u^2 + v^2)/2;
\]

\[
F_1 = U_2, \quad F_2 = U_2 U_3 / U_1, \quad F_3 = \gamma U_2 U_4 / U_1 - (\gamma - 1) U_2 [U_2^2 + U_3^2] / 2 U_1,
\]

\[
G_1 = U_3, \quad G_2 = U_2 U_3 / U_1, \quad G_3 = (\gamma - 1) U_4 + [(\gamma - 1) U_2^2 - (\gamma - 1) U_3^2] / 2 U_1,
\]

\[
G_4 = \gamma U_3 U_4 / U_1 - (\gamma - 1) U_3 [U_2^2 + U_3^2] / 2 U_1.
\]

By considering \( (x, y, t) \) as coordinates of a three-dimensional Euclidean space \( E_3 \) and using Gauss' divergence theorem, it follows that Eq. (1) is equivalent to the following integral equation:

\[
\int_{S(V)} H_m \cdot dS = 0, \quad m = 1, 2, 3, 4, \tag{2}
\]

where \( S(V) \) denotes the surface area around a volume \( V \) in \( E_3 \) and \( H_m = (F_m, G_m, U_m) \), \( m = 1, 2, 3, 4. \)
2.2 Unstructured CE/SE

The CE/SE scheme is naturally adapted to unstructured triangle grids. Each triangle center (O in $\Delta ABC$) and its 3 neighboring triangle centers ($D, E, F$) form 3 conservation elements or CEs (quadrilateral cylinders – $ABDO$, $BECO$ and $CFAO$), as shown in Fig. 1. These triangle centers are the nodes where the unknowns $U$, $U_x$, $U_y$ are defined.

Assume that at the previous time level $n$ (Fig. 1), $U$, and its spatial derivatives $U_x$, $U_y$ are given at all the nodes (triangle centers), the CE/SE time marching is based on a 'tripod' mode, i.e., $U$, $U_x$, $U_y$ at the new time level $n+1$ (e.g., the shaded circle $O'$ in Fig. 1) are computed from their data at its surrounding neighboring nodes at time level $n$ (e.g., $D$ in Fig. 1, note that $O$ itself is not used).

During the time marching, the above flux conservation relation (2) in space-time is the only mechanism that transfers information between node points. A conservation element $CE$, or computational cell, is the finite volume to which (2) is applied. Discontinuities are allowed to occur in the interior of a conservation element and nowhere else. A solution element $SE$ associated with a node (e.g., $D, E, F$ in Fig. 1) is here a set of interface planes in $E_3$ that passes through this node (e.g., planes $DA/A'D', DB/B'D'$, $DC/C'O$ associated with node $D$). Within a given solution element $SE(j,n)$, where $j, n$ are the node index, and time step respectively, the flow variables are not only considered continuous but are also approximated by linear Taylor expansions:

$$U^*(x, y, z; j, n) = U^n_j + (U_x^n)_j (x - x_j) +$$

$$(U_y^n)_j (y - y_j) + (U_t^n)_j (t - t^n),$$

$$F^*(x, y, z; j, n) = F^n_j + (F_x^n)_j (x - x_j) +$$

$$(F_y^n)_j (y - y_j) + (F_t^n)_j (t - t^n),$$

$$G^*(x, y, z; j, n) = G^n_j + (G_x^n)_j (x - x_j) +$$

$$(G_y^n)_j (y - y_j) + (G_t^n)_j (t - t^n),$$

where $j$ is the node index of $D, E$ or $F$, the partial derivatives of $F$ and $G$ can be related to the corresponding one of $U$ by using the chain rule and $U_t$ can be obtained from (1). These Taylor expansions are used to accurately evaluating fluxes on $SE'$s e.g., on planes $DA/A'D', DB/B'D'$ or $DC/C'O$.

In principle, the number of equations derived from these flux conservation laws (each $CE$ provides 4 scalar equations and the total number of scalar equations is 12) matches the number of unknowns (here 12 scalar unknowns). All the unknowns are solved for based on these relations. No extrapolations (interpolations) across a stencil of cells are needed or allowed.

In practice, there is no need to solve a $12 \times 12$ equation system. In the space-time $E_3$ space, (2) is applied to the hexagon cylinder $ABDECF - A'D'B'E'C'F'$. $U$ at the hexagon center at the new time level is first evaluated and then $U$ at the center $O'$ of triangle $A'B'C'$ is obtained by Taylor expansion. Since the evaluation of $U_x$ and $U_y$ involves application of artificial damping and some form of limiters (weighted average), the reader is referred to the original papers [1-3] of Chang et al for details.

Figure 1: CE/SE unstructured grid, showing 3 CEs and the hexagon cylinder in $E_3$.

2.3 Non-Reflecting Boundary Conditions

In the CE/SE scheme, non-reflecting boundary conditions (NRBC) are constructed so as to allow fluxes from the interior domain to smoothly exit to the exterior of the domain. Some variants of the NRBC frequently employed in CE/SE schemes are:

(1) Type 1 - 'steady NRBC':

For a ghost grid node $(j, n)$ lying outside the domain at the top (or bottom) of the domain the NRBC requires that

$$(U_x^n)_j = (U_y^n)_j = 0,$$

while $U^n_j$ is kept fixed at the initially given steady boundary value.
(2) Type II - 'outflow NRBC':

At the downstream boundary, where there are substantial gradients in y direction, the NRBC requires that

\[ (U_x)_{j'}^n = 0, \quad U_j^n = U_{j'}^{n-1/2}, \quad (U_y)_{j'}^n = (U_y)_{j'}^{n-1/2}, \]

where \( j' \) is the index of an interior node closest to the boundary ghost node \( j \) and \( U_j^n \) and \( (U_y)_{j'}^n \) are now defined by simple extrapolation from the interior. This NRBC is valid for either supersonic or subsonic flows. It should be noted that although these NRBC's bear similarity to those used in finite difference schemes, the role they play is very different. It can be shown that the above NRBC's allow fluxes to smoothly exit to the exterior of the domain.

In the following sections, the 2-D CE/SE Euler (Navier-Stokes) scheme is tested to demonstrate its capability and robustness for aeroacoustics computations. Several selected problems in linear and nonlinear aeroacoustics computations are presented. The numerical results, which cover a wide spectrum of waves, from linear and nonlinear acoustic waves to discontinuous waves (shocks), are then compared to available exact solutions or experimental ones.

3 2-D Linear Aeroacoustics Test Problems

Three typical linear aeroacoustics examples are considered in this section. More details can be found from the appropriate references.

3.1 Acoustic Pulse, Entropy Wave, and Vorticity Wave Propagation

This problem [5] is one of the benchmark problems of the first CAA Workshop (Category 3, Problem 1) [4]. The computational domain in the \( x-y \) plane is a square with \(-100 \leq x \leq 100, \) and \(-100 \leq y \leq 100.\) A uniform \( 201 \times 201 \) grid is used with \( \Delta x = \Delta y = 1. \) Initially, a Gaussian acoustic pulse is located at the center of the domain \((x = y = 0)\) and a weaker entropy/vorticity disturbance is located off center \((x = 67, y = 0)\), with a mean flow of Mach number \( M = 0.5.\) The corresponding non-dimensional pressure, density, and streamwise and transverse velocity components are given by

\[ p = \frac{1}{\gamma} + \delta e^{-\alpha_1(x^2+y^2)}, \]
\[ \rho = 1 + \delta [e^{-\alpha_1(x^2+y^2)} + 0.1\delta e^{-\alpha_2[(x-67)^2+y^2}]], \]
\[ u = M + 0.04\delta ye^{-\alpha_2[(x-67)^2+y^2]}, \]
\[ v = -0.04\delta (x-67)e^{-\alpha_2[(x-67)^2+y^2]}, \]

where \( \alpha_1 = \ln 2/9, \) \( \alpha_2 = \ln 2/25, \) and \( \delta \) is an amplitude factor. By choosing a small \( \delta = 0.001,\) the Euler equations are practically linearized. During the computation, the NRBC of Type-I described above is enforced at all boundaries. Fig. 2 illustrates the density contours at different time steps. It also shows that the simple NRBC is effective. Comparison of perturbed density distributions along the centerline \( y = 0 \) with the exact solutions at \( t = 60 \) and \( t = 100 \) are demonstrated in Plots (a) and (b) respectively. The numerical results agree well with the theoretical ones, with no visible dispersion error.

3.2 Linear Instability of a Free Shear Layer and Vortex Roll-Up

In the 2nd example [5], the inviscid linear and nonlinear instability properties of a free shear layer is studied. The numerical results are compared with linear results obtained by the normal mode approach.

The background mean flow consists of a fast (supersonic) stream on the top half domain and a slow (subsonic) stream at the bottom half, with the two parallel streams connected by a shear layer of hyperbolic tangent shape. The non-dimensional flow states are given as:

\[ U_1 = 1, V_1 = 0, \rho_1 = 1/3.15, \quad \rho_1 = 1, \]
\[ U_2 = 0.7391304, V_2 = 0, \rho_2 = 1/3.15, \rho_2 = 0.5405405. \]

The computational domain spans between \( 0 \leq x \leq 300 \) and \(-10 \leq y \leq 10,\) with a uniform structured grid of \( 601 \times 101 \) \((1201 \times 201 \) for fine grid), with time step size \( \Delta t = 0.15.\) The computation is carried out until \( t = 390 \) when the spatial instability is fully developed. A small harmonic perturbation (amplitude \( \delta = 0.001 \) ) at the most unstable frequency \( f = 0.062 \) is enforced at the inlet boundary. Fig. 3 shows the power spectrum \( P_n \) of the computed
Figure 3: Power spectra at $x = 50, 100, 150$, and $250$ with forcing at the most unstable frequency according to linear theory; (coarse grid)

$u'$ (perturbed $u$) in natural log scale at the streamwise stations $x = 50, 100, 150$, and $250$, which correspond to about $3.5, 7$ wavelengths and so on. At $x = 50$, there is a clearly discernable peak centered about the forcing frequency $f = 0.062$. At about $x = 100$, second and third harmonics start to emerge. Further downstream, more harmonics appear and eventually the fundamental saturates. The streamwise evolution of the disturbance amplitude along the horizontal centerline $y = 0$ is illustrated in Fig 4. Numerical results for both the coarse and fine grids are presented and compared to the theoretical linear growth. It is seen that for $0 \leq x \leq 100$ (about $7$ wavelengths) the coarse-grid and fine-grid computations yield good agreement both between themselves and with the linear result. Further downstream, nonlinear effects become important, the growth rate is reduced from the linear value, and ultimately the fundamental saturates. Both numerical results agree reasonably well with each other also in this nonlinear region. Fig. 5 shows the streamwise evolution of the disturbance phase. The numerical results for both the coarse and finer grids are compared to the corresponding result from linear theory. The agreement is surprisingly good until well into the nonlinear region. Fig. 6 compares the normalized $|u'|$ profile across the shear layer flow with the eigenfunction from linear stability theory at the streamwise station $x = 100$, which is located towards the end of the linear growth region. The agreement is excellent. The phase variation across the shear layer of the disturbance at the same station is depicted in Fig. 7. Fig. 8 shows the contours of the flow variables for the finer-grid computation. This figure clearly demonstrates the effectiveness of
the NRBC's at the top, bottom and outlet. It should be emphasized that the domain shown in the figures is exactly the computational domain, no buffer zones, cut-offs or other numerical fixes are applied.

3.3 Mach radiation of a fully expanded axisymmetric jet

Another interesting linear or quasi-linear wave phenomenon is the Mach radiation from a fully expanded supersonic jet. In the following examples, a perturbation (or forcing) is provided by a right-hand-side source term in the energy equation, the 4th component of Eq. (1), located at the origin \((0,0)\) inside the jet core:

\[
\frac{\delta}{\gamma - 1} \exp[-B(x^2 + y^2)] \cos(\omega t),
\]

where \(\delta\) is a small number \((0.00005 < \delta < 0.001)\), \(\omega = 2\pi St\), and the constant \(B = 8\).

3.3.1 Mach radiation from a supersonic axisymmetric jet

In this test example [6], a fully expanded supersonic jet with Mach number \(M_j = 2.0\) is considered. The computational domain spans between \(0 \leq x \leq 33D\) and \(0 \leq y \leq 19D\), with a non-uniform structured grid of \(350 \times 281\) nodes, where \(D\) is the diameter of the jet at the nozzle exit. More grid points are packed around the shear layer. The perturbation source strength \(\delta = 0.001\). The computation of the unsteady jet flow is carried out to \(t = 100\) when the spatial instability is fully developed. Fig. 9 illustrates the isobars and v-velocity contours in the near and intermediate field. The Mach radiation flow pattern agrees qualitatively with experimental [14] and other computational [15] results.

3.3.2 Mach wave system inside the jet \((M=1.2)\)

It was found that, as analyzed by Tam and Hu [7], the Mach radiation attenuates exponentially outside the jet shear layer, while the Mach wave system is trapped inside the jet and forms a cross-hatch pattern. Here the jet shear layer plays the role as a 'wave guide'.

The computational domain is \(0 \leq x \leq 3D\) and \(0 \leq y \leq 1D\), with a non-uniform grid of \(331 \times 101\) nodes, where again \(D\) is the diameter of the jet at the nozzle exit. The perturbation source strength \(\delta = 0.00005\), with a high Strouhal number \(St = 2.0\). A time step size \(\Delta t = 0.0015\) is chosen. The computation of the unsteady jet flow is carried out for 12000 steps to ensure that the spatial instability is fully developed.

Fig. 10 clearly demonstrates the cross-hatch pattern of the Mach wave system inside the 'wave guide'—the jet core. The Mach waves are repeatedly reflected because they do not penetrate the annular shear layer. The pat-
tern qualitatively agrees with the analysis of Tam and Hu [7]. At a low supersonic Mach number $M_j = 1.2$, at the outer surface of the shear layer, the local phase velocity can hardly exceed the local speed of sound and hence practically no exterior Mach radiation occurs.

4 2-D Non-linear Aeroacoustics Test Problems

In this section, two examples of non-linear aeroacoustics involving shocks are considered. For the first example, more details can be found from [5].

4.1 Multiple Interaction of a Strong Vortex and Shocks

Vortex-shock interaction has been considered a ‘difficult’ problem for CAA. However, the CE/SE Euler scheme can solve the problem in an effortless way, without any numerical trick or fix [5]. As seen in Fig 11, a uniform structured grid of $401 \times 101$ nodes is employed in the rectangular domain with $\Delta x = \Delta y = 1$. The inflow boundary condition is given as a supersonic flow with a Mach number of 2.9:

$$u_0 = 2.9, \quad v_0 = 0, \quad p_0 = 1/1.4, \quad \rho_0 = 1.$$ 

The boundary condition at the top is an inclined flow:

$$u_t = 2.6193, \quad v_t = -0.50632, \quad p_t = 1.5282, \quad \rho_t = 1.7000.$$ 

The outflow boundary condition is the type-II NRBC and the bottom is a solid reflecting wall. Then, a steady oblique shock is formed with $29^\circ$ inclination and reflected at the bottom wall. The flow with shocks is pre-calculated until a steady state is reached. It is then used as the background mean flow for further computation.

At $t = 0$, a strong Lamb’s vortex is placed at $x = 22, y = 60$. With $\Delta t = 0.2$, 900 time steps were run.

4.2 Screech Tone Noise of an Underexpanded Jet

In nature, the screech noise is caused by an acoustic feedback loop between the jet nozzle exit lip (or nearby solid surfaces) and the shock cell structure. At the low supersonic jet Mach number $M_j = 1.19$, the overall motion in the experiment [8] is in an axisymmetric mode, and the use of axisymmetric 2-D scheme is appropriate.

The Navier-Stokes CE/SE solver with LES (large eddy simulation) is applied to a triangular unstructured grid of 88300 elements (cells). The geometry of the computational domain is shown in Fig 12. Both Type I and II
NRBC's are applied appropriately to all boundaries except at the nozzle solid walls where no-slip solid wall boundary condition is employed, and at the nozzle exit where the following inflow condition is applied:

\[ p_e = p_0 \left[ \frac{1 + \frac{1}{2}(\gamma - 1)M_j^2}{1 + \frac{1}{2}(\gamma - 1)} \right]^{\frac{-\gamma}{\gamma-1}} \]

\[ \rho_e = 0.5(\gamma + 1)p_0 \left[ \frac{1 + \frac{1}{2}(\gamma - 1)M_j^2}{1 + \frac{1}{2}(\gamma - 1)} \right]^{\frac{-\gamma}{\gamma-1}} \]

\[ u_e = \left( \frac{2}{\gamma + 1} \right)^{1/2}, v_e = 0. \]

Initially the entire field is set to the ambient flow:

\[ \rho_0 = 1, \quad u_0 = 0, \quad v_0 = 0, \quad p_0 = \frac{1}{\gamma}. \]

In the numerical simulation, the initial impact of the boundary condition at the nozzle exit will stimulate the jet shear layer and trigger the feedback loop to form the self-sustaining oscillation that generates screech waves.

Figure 13: Schlieren picture from experiment (Panda), showing shock cells.

Fig. 14 illustrates the radiating screech waves at the time level of 212,800 steps. Since there is no forcing at all, these waves are a sure sign of the presence of a sustainable self-excited oscillation. In the processing of the numerical data for this figure, the very high level isobar contours, corresponding to hydrodynamic waves around the jet core area are 'cut off' and the 'colors' are appropriately adjusted so that the acoustic waves are clearly displayed. The screech wavelength is about 1.6D, well in line with experimental results (e.g., [8]) and theoretical predictions (e.g., Seiner [10], Tam [9]). The shock cell structure in Fig. 14 is comparable to the experimental Schlierens in Fig. 13. Fig. 15 shows the PSD (power spectrum density) at the location \((x = 2.0, y = 6.0)\). It is seen that the spikes at the fundamental frequency and the subharmonics shoot out against the background noise, despite the low resolution. The fundamental frequency is estimated to be about 8,661 Hz in this case, comparable to the experimental case - 8,425 Hz.

5 The Airframe Noise Problems

Airframe noise is an important noise source in aeroacoustics. Several airframe tone noise problems are considered in this section to demonstrate the capability of the CE/SE scheme.

5.1 The subsonic cavity noise problem

In this problem[11], a \( M = 0.8 \) subsonic flow passes over a cavity of aspect ratio 6.5. Tonal oscillations occur in a feedback cycle in which the vortices shed from the upstream lip of the cavity convect downstream and impinge on the other lip, generating acoustic waves that in turn propagate upstream to excite new vortices (Fig 16).

In the computational domain, 42,000 triangular elements are used in the unstructured grid, which is made from structured rectangular cells. Figure 17 depicts with isobars where the acoustic waves are generated and propagate in a series of snapshots (1-12) in the near field of the cavity. Each snapshot is 3.6 (720 steps) unit apart in time. Initially, the flow conditions are set to be identical to the ambient flow. The boundary conditions are the appropriate types of NRBC except at the cavity walls, where slip, reflective wall boundary condition is applied. No visible reflections are observed at the non-reflecting bound-
Figure 15: PSD at \((x = 2.0, y = 6.0)\); x-axis: Strouhal number.

Figure 16: Sketch of the subsonic cavity noise problem.

aries. At the inflow boundary, upstream propagating waves are well absorbed within 2 cells, without contaminating the interior domain. From an animation of the solution, the near field acoustic wave structure appears to be complicated and chaotic.

5.2 Auto-door gap noise

Figure 18 illustrates the geometric configuration of the door gap noise problem, which represents a typical cavity aeroacoustic feedback system. The problem is given as a benchmark problem at the 3rd CAA Workshop [12] (OAI/NASA Glenn, Cleveland, OH, November, 1999). \(2C\) (17.52 mm) is chosen as the length scale and the speed of sound and density of the ambient flow are respectively the scales for velocity and density. In the current computation, there are 48,184 triangle elements for the unstructured mesh in the computational domain. These triangles are actually obtained by dividing a rectangular structured mesh cell into 4 pieces. The rectangular cell keeps a uniform size of \(\Delta x = 0.00625\) and \(\Delta y = 0.01136\) around the area of the gap and the interior of the cavity, but grows larger near the boundaries. The mean flow follows the \(x\)-direction with Mach number \(M = 0.1497\), corresponding

Figure 17: Isobar snapshots of the subsonic cavity flow field.

Figure 18: Sketch of the auto-door gap noise problem with actual sizes.

Figure 19: Isobar snapshots at various time steps in the flow field, showing formation of vortices.
to the car speed of 50.9 m/s. Initially, ambient flow condition is imposed on the entire flow field. At the inflow, the $M = 0.1497$ flow is imposed. At the top and bottom the Type I NRBC is used while at the outflow, the Type II NRBC is specified. The domain shown in Fig. 19 is exactly the computational domain, no buffer zone is used but still, the CE/SE NRBC is very effective. The generation of vortices and the generation of (nonlinear) acoustic waves by vortex-gap edge impingement, both inside and outside of the cavity, are clearly displayed. The computation is totally carried out for 225,000 time steps. Starting from time step 60,000, when the unsteady flow is considered as fully developed, the pressure history at the mid-point of the left wall of the cavity is recorded and provides for FFT (fast Fourier transform) analysis. Fig. 20 displays the PSD (power spectrum density) in log. scale for the time series. The $x-$axis denotes the reduced frequency – the Strouhal number $St$. $St = 1$ is equivalent to a frequency of 19,406 Hz. It is observed that there are several different tone spikes extending to very high frequencies. The frequency at the lowest tone spike is at 1839 Hz. This agrees well with the experimental frequency data (1824 Hz) provided by the 3rd CAA Workshop.

5.3 Blunt trailing edge noise

In this last example, the noise generated by the von Karman vortex street downstream of a rectangular blunt slab is computed. The flow Mach number is $M = 0.3$. Figure 21 illustrates the details of the computational domain. Initially, the entire field is set at the ambient flow. Then the inflow of $M = 0.3$ is imposed at the left boundaries. At the solid slab walls, no-slip condition is applied. The computation is totally carried out for 390,000 time steps. Time history at the data point as shown in Fig. 21 is recorded for FFT analysis. The power spectrum density (PSD) is shown in Fig. 22. As before, the $x-$axis denotes the reduced frequency – the Strouhal number $St$. $St = 1$ is equivalent to a frequency of 13,386 Hz if the thickness of the slab is assumed to be one inch. The frequency at the lowest tone spike is 784 Hz. Although there is currently no experimental data known to the author, this frequency doesn’t seem to be in conflict with other experiments, e.g., a $M = 0.2$ flow past a circular cylinder of diameter 1.9 cm, with the frequency of $f = 643$ Hz (the 2nd CAA Workshop, Category 4 [13]).

6 Concluding remarks

Through numerous numerical examples, the capability of the new CE/SE schemes is demonstrated. The CE/SE scheme is a time-accurate conservative scheme for CFD and CAA. Its features include:

1. naturally adapted to unstructured grid,
(2) high resolution, low dispersion, and low dissipation, despite its nominal 2nd order accuracy (in space and time),
(3) robust, treats the 'difficult' problems in a simple, effortless way, particularly appropriate for near field, non-linear aeroacoustics,
(4) the novel NRBC is simple, effective and truly multi-dimensional.

In the near future, 3-D LES or turbulence modeling will be considered in the CE/SE scheme development.

References


Computational Aeroacoustics by the Space-Time CE/SE Method

Ching Y. Loh

Taitech, Inc.
21000 Brookpark Road
Cleveland, Ohio 44135

National Aeronautics and Space Administration
Washington, DC 20546-0001


In recent years, a new numerical methodology for conservation laws—the Space-Time Conservation Element and Solution Element Method (CE/SE), was developed by Dr. Chang of NASA Glenn Research Center and collaborators. In nature, the new method may be categorized as a finite volume method, where the conservation element (CE) is equivalent to a finite control volume (or cell) and the solution element (SE) can be understood as the cell interface. However, due to its rigorous treatment of the fluxes and geometry, it is different from the existing schemes. The CE/SE scheme features: (1) space and time treated on the same footing, the integral equations of conservation laws are solved for with second order accuracy, (2) high resolution, low dispersion and low dissipation, (3) novel, truly multi-dimensional, simple but effective non-reflecting boundary condition, (4) effortless implementation of computation, no numerical fix or parameter choice is needed, and (5) robust enough to cover a wide spectrum of compressible flow: from weak linear acoustic waves to strong, discontinuous waves (shocks), appropriate for linear and nonlinear aeroacoustics. Currently, the CE/SE scheme has been developed to such a stage that a 3-D unstructured CE/SE Navier-Stokes solver is already available. However, in the present paper, as a general introduction to the CE/SE method, only the 2-D unstructured Euler CE/SE solver is chosen as a prototype and is sketched in Section 2. Then applications of the CE/SE scheme to linear, nonlinear aeroacoustics and airframe noise are depicted in Sections 3, 4, and 5 respectively to demonstrate its robustness and capability.