COMPUTATIONAL AEROACOUSTIC ANALYSIS SYSTEM DEVELOPMENT

A. Hadid, W. Lin, E. Ascoli, S. Barson, and M. Sindir

Rocketdyne Propulsion & Power

The Boeing Company

Canoga Park, CA 91309-7922

ABSTRACT

Many industrial and commercial products operate in a dynamic flow environment and the aerodynamically generated noise has become a very important factor in the design of these products. In light of the importance in characterizing this dynamic environment, Rocketdyne has initiated a multiyear effort to develop an advanced general-purpose Computational Aeroacoustic Analysis System (CAAS) to address these issues. This system will provide a high fidelity predictive capability for aeroacoustic design and analysis. The numerical platform is able to provide high temporal and spatial accuracy that is required for aeroacoustic calculations through the development of a high order spectral element numerical algorithm. The analysis system is integrated with well-established CAE tools, such as a graphical user interface (GUI) through PATRAN, to provide cost-effective access to all of the necessary tools. These include preprocessing (geometry import, grid generation and boundary condition specification), code set up (problem specification, user parameter definition, etc.), and postprocessing.

The purpose of the present paper is to assess the feasibility of such a system and to demonstrate the efficiency and accuracy of the numerical algorithm through numerical examples. Computations of vortex shedding noise were carried out in the context of a two-dimensional low Mach number turbulent flow past a square cylinder. The computational aeroacoustic approach that is used in CAAS relies on coupling a base flow solver to the acoustic solver throughout a computational cycle. The unsteady fluid motion, which is responsible for both the generation and propagation of acoustic waves, is calculated using a high order flow solver. The results of the flow field are then passed to the acoustic solver through an interpolator to map the field values into the acoustic grid. The acoustic field, which is governed by the linearized Euler equations, is then calculated using the flow results computed from the flow solver.

NOMENCLATURE

c	Speed of sound
f	forcing function
L	characteristic length
M	Mach number
m	number of equations

p   pressure

Re   Reynolds number

t non-dimensional time ($=L/c$)

$\underline{u}$ solution vector

u, v   velocities in x and y-directions respectively

x, y   Cartesian coordinates

ρ density

δt time step

INTRODUCTION

Many aerospace and commercial products are operated in a dynamic flow environment. The structural integrity, performance and development costs of these products are affected by the unsteady flow fields they encounter. In a rocket propulsion system, dynamic loads are attributed as the cause of many life limiting and failure mechanisms. Unsteady flows can also be a very effective sound generating mechanism; George [1] states that the aerodynamically generated noise increases approximately as velocity to the 6th power. Sound may be generated whenever a relative motion exists between two fluids or between a fluid and a surface. Examples of flow-induced noise in an aerospace or automotive environment are numerous. Airplanes, helicopters, jet engines, turbomachinery and rockets all exhibit undesirable noise characteristics. In all these applications the common physical processes that are responsible for noise generation include turbulent fluid motion, structural vibration and unsteady aerodynamics. This coupled unsteady fluid dynamics and acoustic environment is poorly understood and engineering tools are required to analyze this phenomenon. In light of the importance in characterizing the dynamic fluid-acoustic environment, Rocketdyne has initiated a multi-year effort to develop a general-purpose computational fluid dynamics based analysis system for dynamic fluid-acoustic prediction. This is a fully integrated system that will provide high fidelity predictive capability through the development of a novel and accurate numerical algorithm. The numerical algorithm is a high order method based on the least squares spectral element method (LSSEM), which provides the required capability to accurately model complex geometries and rapidly varying flow and acoustic fields.

In what follows, we describe the coupled computational aeroacoustic system that is developed and the numerical method used with a demonstration of the accuracy and effectiveness of this method.

CAAS SYSTEM DEVELOPMENT

CAAS is an integrated general-purpose analysis tool for aeroacoustic engineering and design. It is intended to provide a detailed temporal description of an acoustic field in terms of signal intensity, frequency distribution, and propagation pattern. Numerical simulation of an acoustic field generally requires high temporal and spatial resolution. CAAS employs a highly accurate numerical algorithm based on the least-squares spectral element method. The code developed, Unstructured implicit Flow solver (UniFlo), is a high-order accurate numerical platform needed for acoustics calculations and is an expanded version of the original algorithm proposed by Chan [2]. It can accurately resolve the acoustic effects at the near and mid fields that are covered with a suitable mesh. The CAAS system consists of five basic components: a GUI, a preprocessor, an analysis code, a postprocessor, and a data base management system (DBMS). At the heart of the system is the analysis code that will actually carry out the aeroacoustic calculations. The other four components interface and work efficiently with the analysis code. The GUI provides the primary interface for users to interact with the system and perform the necessary code setup (boundary and initial conditions,

etc.). The pre- and postprocessors define the design geometry, construct the computational model to facilitate analysis, and graphically present the results of the calculations. The DBMS manages the various system databases and provides data access. The commercial software PATRAN from the MacNeal-Schwendler Corporation (MSC) is used as the platform for the GUI, pre-and postprocessing, and DBMS system modules. The numerical algorithm will be described in details in section 3. However, the other major system level features are described below.

GRAPHICAL USER INTERFACE (GUI)

The GUI is constructed to present the user with information needed to perform operations, during any step of the analysis cycle. The system will prompt the user for inputs to setup initial and boundary conditions for example. It incorporates a high degree of logic to prompt the user for inputs or it will fill relevant default values when none is selected. The system will also guide the user toward a faster setup and better code performance. There is also a set of tutorials for new users to get started through sample problems in a step by step fashion. A 'help' button is also available to provide the user with relevant information for the task at hand and access to online manuals.

PREPROCESSING

Preprocessing includes all steps needed to define the problem to be solved prior to executing the computational solver. This includes the ability to import or create the geometric description, generate a suitable computational mesh, and provide the required code inputs. The standard PATRAN capability has been enhanced to generate spectral elements of arbitrary order from finite elements (quads and hex) using special translator and grid generation tools. These tools support conversion from quadrilateral and hexahedral elements to a conforming spectral element grid of arbitrary order. Graphical display tools are provided to visualize both the geometry and the mesh generated together with the capability to manipulate and change the geometry and mesh for effective modeling of the design hardware. GUI panels also include code setup parameters such as fluid properties, control inputs and boundary conditions.

POSTPROCESSING

CAAS provides the user with the capability to extract and visualize results in different forms, from simple line plotting to 3D surface and vector plots. Transient flow visualization with animation schemes can also be employed. Graphics may be stored or exported in different forms, such as postscript, RGB, raster, gif or tif.

DATABASE MANAGEMENT SYSTEM (DBMS)

The CAAS database contains all of the preprocessing information (grid, setup parameters, boundary and initial conditions), solution files (stored at specified time intervals), and postprocessed information for both the flow and acoustic fields. The approach is to utilize existing PATRAN database which includes basic geometry information and other code setup together with a second database that is specific to aeroacoustic solutions. The spectral order computational grid will be stored in this database along with the solution variables at each specified time step. The two databases are linked and will appear as one to the user.

Other nongraphical system features are also available and are needed for effective use in a real design environment. These include "session file" that saves all meaningful keystrokes and mouse interactions

which will allow the user to quickly recover the work completed during a working session in case of an unplanned interruption (e.g. power failure) without repeating all of the commands. The system also allows the user to create a script including all desired operations and then run the script by executing a single command. Templates for a general class off problems can also be setup to avoid many redundant steps in the problem setup.

NUMERICAL METHOD

The numerical platform for both the flow and acoustic solvers is based on the least-squares spectral element method (LSSEM) operating exclusively in physical space in order to handle complex geometry and a variety of boundary conditions. The method has low dispersive and dissipative errors which renders it ideal for predicting signal propagation such as acoustic waves. The unsteady fluid motion which is governed by the incompressible Navier-Stokes equations are responsible for both the generation and propagation of acoustic waves which are governed by the linearized Euler equations derived from the perturbation expansion about Mach number in the subsonic regime. The LSSEM is an extension of the finite element method proposed by Jiang et al. [3] who cast the governing equations as a set of first order system as

$$L\vec{u} = \vec{f} \tag{1}$$

L is a first order partial differential operator given as;

$$L\vec{u} = \sum_{i=1}^{n_d} A_i \frac{\partial \vec{u}}{\partial x_i} + A_0 \vec{u} \tag{2}$$

$n_d = 2$ or 3, depending on the spatial dimensions, x_i's are the Cartesian coordinates, $\vec{u}$ is an n-dimensional solution vector of the dependent variables, $\vec{f}$ is the forcing function. A's are m x n matrices, which describe the characteristics of the system of equations being solved. m is the number of equations in the system. The idea behind LSSEM is to minimize the residual function, $\vec{R}$ such that $\vec{R} = L\vec{u} - \vec{f}$ and construct a least squares functional as

$$I(\vec{u}) = \frac{1}{2}\int_{\Omega}(L\vec{u} - \vec{f})^2 d\Omega \tag{3}$$

Isoparametric mapping is employed to transform the equations from the Cartesian (x,y) system to a generalized (ξ,η) coordinate system (in two-dimensions for example). The dependent variable $\vec{u}$ is approximated as;

$$u(\xi_l,\eta_m) = \sum_{i}^{M}\sum_{j}^{M}\Psi_i(\xi_l)\Phi_j(\eta_m)\{a_{ij}\} \tag{4}$$

Where $\Psi_i(\xi_l)$ and $\Phi_j(\eta_m)$ are one-dimensional linearly independent shape functions. $\{a_{ij}\}$ are the unknown expansion coefficients for the dependent variables and M is the total number of basis functions (or degree of freedom) in each direction of an element. The basis functions used in the present study are the Lagrangian interpolant based on Legendre polynomials of the independent variables. Substituting Eq. (4) into Eq. (1), forming the residual and applying the method of least squares [4] with respect to the expansion

coefficients and using Gaussian quadrature for numerical integration leads to a set of algebraic equations. These equations are solved by the conjugate gradient method with Jacobi preconditioner.

For Euler equations, the working variables are density, velocities and pressure. For brevity, one can write the equations in the two-dimensional Cartesian coordinate system as;

$$
\begin{bmatrix}
\alpha_1 + \delta t M_\infty \dfrac{\partial}{\partial x} & \delta t \dfrac{\partial}{\partial x} & \delta t \dfrac{\partial}{\partial y} & 0 \\[3mm]
0 & \alpha_1 + \delta t M_\infty \dfrac{\partial}{\partial x} & 0 & \delta t \dfrac{\partial}{\partial x} \\[3mm]
0 & 0 & \alpha_1 + \delta t M_\infty \dfrac{\partial}{\partial x} & \delta t \dfrac{\partial}{\partial y} \\[3mm]
0 & \delta t \dfrac{\partial}{\partial x} & \delta t \dfrac{\partial}{\partial y} & \alpha_1 + \delta t M_\infty \dfrac{\partial}{\partial x}
\end{bmatrix}
\begin{bmatrix} \rho \\ u \\ v \\ p \end{bmatrix}
=
\begin{bmatrix}
-\alpha_2 \rho^n - \alpha_3 \rho^{n-1} \\
-\alpha_2 u^n - \alpha_3 u^{n-1} \\
-\alpha_2 v^n - \alpha_3 v^{n-1} \\
-\alpha_2 p^n - \alpha_3 p^{n-1}
\end{bmatrix}
$$

In the above, the coefficients of the convective terms are treated as constant, though in practice, they are determined from the solution to the Navier-Stokes equations. The accuracy is second order in time with the application of a backward differencing scheme for which $\alpha_1 = 1.5$, $\alpha_2 = -2.0$, and $\alpha_3 = 0.5$. UniFlo employs isoparametric mapping to transform the above equation from the Cartesian coordinate system to a generalized coordinate system where the spatial discretization is performed. The spatial accuracy depends on the choice of basis functions and the type of elements used. In UniFlo, quadrilateral (in 2D) and hexahedral elements (in 3D) as well as Legendre polynomial based spectral element developed by Ronquist and Patera [5] are used.

NUMERICAL RESULTS

To evaluate the effectiveness of LSSEM, we apply it to three representative problems. The first two problems highlight the complex interaction of acoustic wave propagation and reflection, while the third problem shows the effects of vortices or wakes shedding from a square cylinder on the propagation of acoustic waves. Although the problems represented here are two-dimensional, CAAS in general has aeroacoustic predictive capability in 2D and 3D curvilinear problems as well.

ACOUSTIC PULSE IN A SEMI-INFINITE DOMAIN

This problem is one of the bench mark test cases suggested in the ICASE/LaRC workshop on computational aeroacoustics [6] to test the effectiveness of wall boundary conditions and the numerical schemes. Figure 1 shows a schematic of an acoustic pulse placed near a rigid wall in a semi-infinite domain. The computational domain used is $0 \le x \le 200$, $0 \le y \le 200$. The wall is at y=0. The linearized Euler equation in two-dimensions are

$$
\frac{\partial}{\partial t}
\begin{bmatrix} \rho \\ u \\ v \\ p \end{bmatrix}
+
\frac{\partial}{\partial x}
\begin{bmatrix} M\rho + u \\ Mu + p \\ Mv \\ Mp + u \end{bmatrix}
+
\frac{\partial}{\partial y}
\begin{bmatrix} v \\ 0 \\ p \\ v \end{bmatrix}
= 0
$$

M is the Mach number = 0.5. The initial condition is

$$\text{at time } t = 0, u = v = 0 \text{ and } p = \rho = \exp\left\{-(\ln 2)\left[\frac{x_0^2 + (y_0 - 25)^2}{25}\right]\right\}$$

where x_0=100, y_0=25 are the coordinates of the initial pulse.

Figure 2 shows time history plots of an acoustic pulse reflected from a hard wall in a uniform flow at Mach 0.5 at times, t = 15, 45, 60, 90, 100 and 150 respectively. The lower boundary is a reflecting wall while the other boundaries are non-reflecting out-going wave boundaries. This problem was used to test the effectiveness of wall boundary condition and out-going wave conditions devised in the acoustic solver. Figure 3 shows a comparison of the computed acoustic pressure with the exact solution along the line $x=y$ at times t=75, 90 and 100 respectively. The computed results show good agreement with the exact solution.

As seen from figure 2, the acoustic pulse expands at the speed of sound and travels with the base flow at Mach 0.5. The downstream wave travels at Mach 1.5, three times faster than the upstream wave, and reaches the downstream boundary early and exits. The pulse reflects from the lower wall, as seen in the red spots, and expands as the original pulse does. Both original and reflected waves continue to expand and travel with the base flow, and then exit the right, left, and upper sides of the domain. The computed results show that both wall reflection and out-going wave conditions work very well, except at t = 150 there is a very tiny reflection of the reflected wave at the downstream boundary.
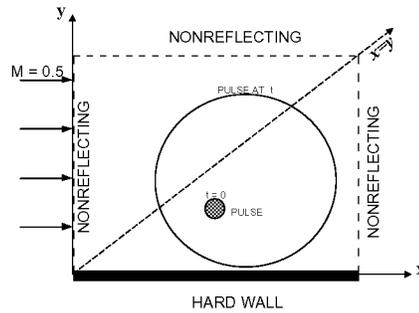

Figure 1. Problem Schematic for an Acoustic Pulse in a Semi-Infinite Domain
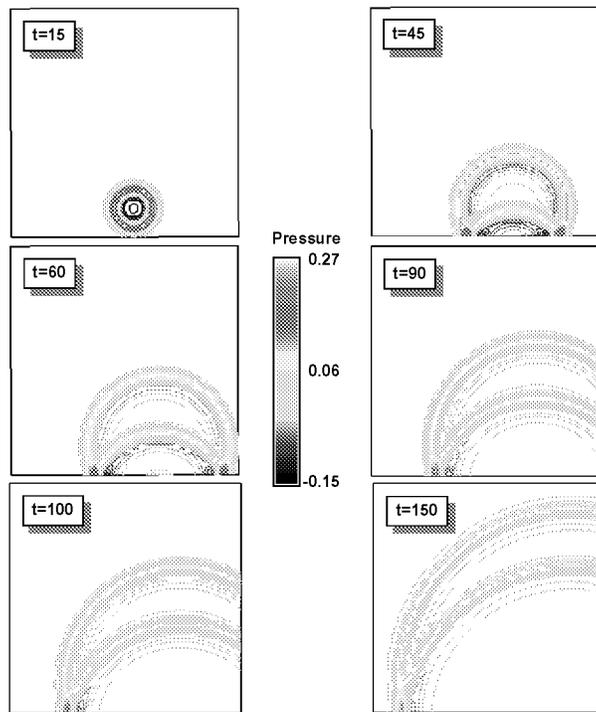
Figure 2. Acoustic pulse propagating in a semi-infinite domain with a uniform flow at Mach 0.5
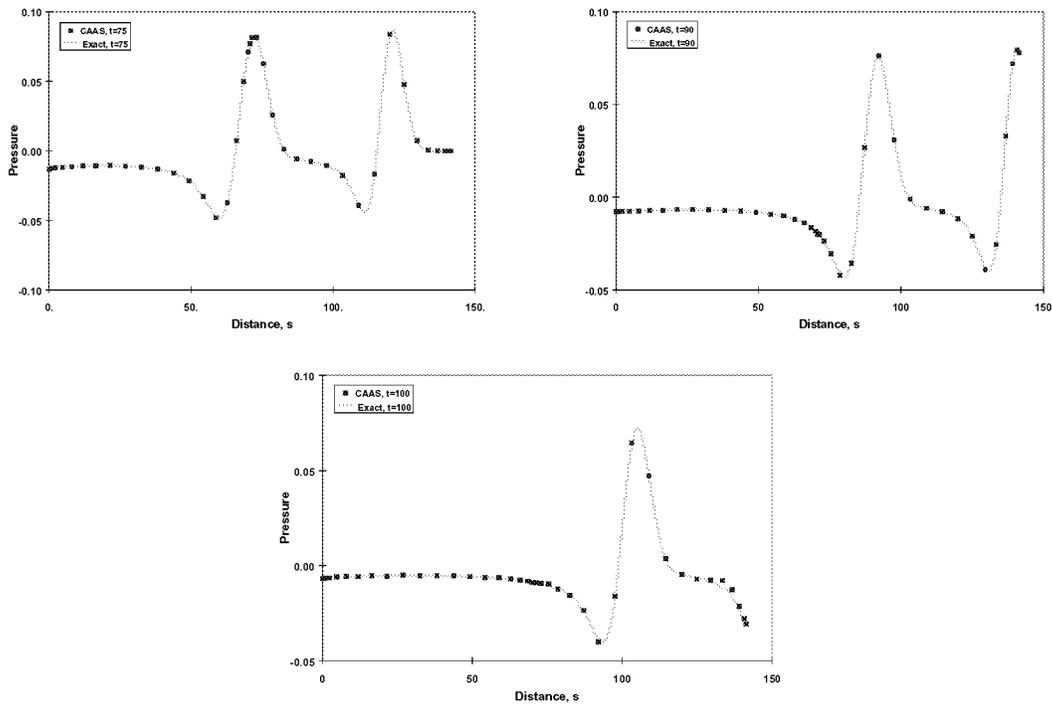

Figure 3. Acoustic pressure along the line $x = y$

ACOUSTIC PULSE IN A DUCT

Figure 4 shows a schematic of an acoustic pulse placed at the center of a duct. Figure 5 shows time history plots of the acoustic pulse traveling in a 2D duct with reflecting walls and a uniform flow at Mach 0.5. This example simulates the propagation of sound waves generated in the inlet and exhaust ducts of an internal combustion engine such as in an automobile. The problem is also used to test the effectiveness of hard wall conditions and nonreflecting (out-going wave) conditions. Initially the sound source is created by some mechanism in the center of the duct. It expands at the speed of sound, travels with the base flow at Mach 0.5, and forms waveguide patterns in the duct.
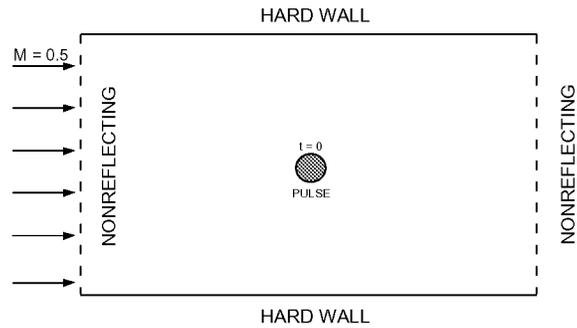

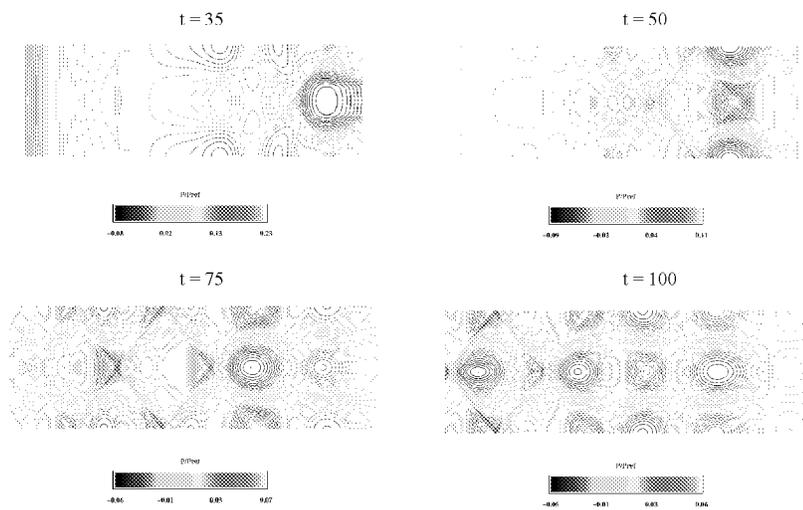Figure 4. Problem schematic for acoustic pulse in a duct

Figure 5. Acoustic pulse propagating in a duct

AEROACOUSTICS OF A 2D VORTEX SHEDDING FROM A RECTANGULAR CYLINDER

The sound generated by a viscous flow past a square cylinder at Re=14,000 is predicted using the linearized Euler equations approach. The time-dependent mean flow is calculated using a separate flow solver that is based on a second order accurate finite-volume method. Time accuracy of the flow solver is assured using the PISO methodology [7], which is essentially noniterative. The solution process is split into a series of steps whereby operations on pressure are decoupled from those on velocity at each time step. The avoidance of iterations substantially reduces the computational effort compared with that required by iterative methods such as in UniFlo.

Calculations are performed for the turbulent flow around a square obstacle of height H in a domain