[2]Peters, N., "Laminar Diffusion Flamelet Models in Non-Premixed Turbulent Combustion," *Progress in Energy and Combustion Science*, Vol. 10, 1984, pp. 319-339.

[3]Liñan, A., "Lewis Number Effects on the Structure and Extinction of Diffusion Flames Due to Strain," *The Role of Coherent Structures in Modelling Turbulence and Mixing*, edited by J. Jimenez, Vol. 136, Lecture Notes in Physics, Springer-Verlag, Berlin, Germany, 1981, p. 333.

[4]Law, C. K., and Chung, S. H., "Steady State Diffusion Flame Structure with Lewis Number Variations," *Combustion Science and Technology*, Vol. 29, Pts. 3-6, 1982, pp. 129-145.

[5]Chung, S. H., and Law, C. K., "Structure and Extinction of Convective Diffusion Flames with General Lewis Numbers," *Combustion and Flame*, Vol. 52, 1983, pp. 59-79.

[6]Seshadri, K., and Trevino, C., "The Influence of the Lewis Numbers of the Reactants on the Asymptotic Structure of Counterflow and Stagnant Diffusion Flames," *Combustion Science and Technology*, Vol. 64, Nos. 4-6, 1989, pp. 243-261.

[7]Hermanson, J. C., and Vranos, A., "Combined Effects of Preferential Thermal and Species Transport in a Strained, Laminar Diffusion Flame," *Combustion Science and Technology*, Vol. 75, Nos. 4-6, 1991, pp.339-345.

[8]Hermanson, J. C., and Vranos, A., "Preferential Thermal and Multicomponent Species Transport Effects in Strained Diffusion Flames with Fast Chemistry," United Technologies Research Center, UTRC Rept. 92-3, East Hartford, CT, April 1992.

[9]Curtiss, C. F., and Hirschfelder, J. O., "Transport Properties of Multicomponent Gas Mixtures," *The Journal of Chemical Physics*, Vol. 17, No. 6, 1949, pp. 550-555.

[10]Mason, E. A., and Saxena, S. C., "Approximate Formula for the Thermal Conductivity of Gas Mixtures," *Physics of Fluids*, Vol. 1, No. 5, 1958, pp. 361-369.

Comparison of Numerical Oblique Detonation Solutions with an Asymptotic Benchmark

Matthew J. Grismer* and Joseph M. Powers†
University of Notre Dame, Notre Dame, Indiana 46556

Introduction

THE design of aerospace vehicles of the present and future must take into account effects of compressibility, shock waves, multidimensionality, chemical reaction zones of various thicknesses, diffusive and radiative transport, and many other physical processes which are present in surrounding fluids. As it is often either prohibitively expensive or physically impossible to test aircraft under flight conditions and because the problems are typically analytically intractable, designers have come to rely on numerical methods to model these phenomena to predict flight performance. To have confidence in a numerical method, it is important to verify that it can reproduce known benchmark analytic solutions, available for simple model problems. A common benchmark is the inert oblique shock solution. Although valuable, this benchmark cannot quantify how well the numerical method predicts the effects of thick reaction zones.

This Note gives a new, more rigorous benchmarking procedure that is appropriate for numerical models of two-dimensional, high-speed reactive flows. The procedure is illustrated by comparing asymptotic and numerical solutions for oblique detonations, defined here as an attached oblique shock followed by an exothermic reaction with a thick reaction zone.

Received Oct. 1, 1991; revision received June 2, 1992; accepted for publication June 5, 1992.

*Center for Applied Mathematics Graduate Fellow, Department of Aerospace and Mechanical Engineering. Student Member AIAA.

†Assistant Professor, Department of Aerospace and Mechanical Engineering. Member AIAA.

The asymptotic solution, valid in the limit of high Mach number M_0, is given by Powers and Stewart,[1] who also give a review of the oblique detonation literature. For tractability, it was necessary in Ref. 1 to consider the fluid to be a calorically perfect ideal gas which undergoes a single-step irreversible exothermic reaction with Arrhenius kinetics in which the reactants and products have the same molecular weight. It was shown in Ref. 1 for such a fluid that an oblique detonation over a straight wedge consists of an attached shock with a downstream reaction and vorticity layer. The shock has maximum curvature at the wedge tip which relaxes to zero far from the wedge tip. Vorticity generated by the shock curvature is convected in a finite layer near the wedge surface. Heat release occurs in a finite layer parallel to the lead shock. Far from the shock and wedge surface the flow relaxes to an irrotational, equilibrium state.

The numerical solution was obtained with the RPLUS code,[2] in development at the NASA Lewis Research Center, using standard available features to simulate the flow. It is not our purpose to address issues which arise from the particular numerical method used in RPLUS. Rather, it is to show the utility of the asymptotic solution of Ref. 1 as a new benchmark. It should be possible in future studies to use this benchmark to critically assess the merits of differing numerical schemes for reacting flows.

As both the asymptotic and numerical solutions are approximations to a presumed exact solution, care must be exercised in determining the conditions under which the asymptotic solutions can be treated as a benchmark. For the second-order central differencing scheme of RPLUS, the numerical error is $\mathcal{O}(\Delta x^2)$ while the asymptotic error is $\mathcal{O}(\epsilon^2)$ (here $\epsilon = 1/M_0^2$). Thus for calculations on a fine fixed grid, one must employ high Mach numbers to insure that the asymptotic solution can be treated as a benchmark relative to the numerical solution. At lower Mach numbers the numerical solution gives a more accurate global estimate, but still contains local inaccuracies near shocks and boundaries. In such cases, the asymptotic solution retains value as a qualitative standard. For the comparisons of this Note, we study a Mach number range of $7.5 \leq M_0 \leq 20$. The asymptotic method, formally valid only when the ratio of chemical energy release to the kinetic energy of the flow is small, breaks down for lower Mach numbers and $\mathcal{O}(1)$ heat release.

We intend this Note to be of use to those developing codes to predict physical results. To this end, our procedure requires a compromise in model complexity in which only limited elements of real gas behavior at high Mach number are retained. Thus, our necessarily simple model ignores the temperature dependency of the specific heat, vibrational relaxation and dissociation effects, diffusive and radiative transport, turbulence, and the multicomponent, multireaction nature of real combustion processes. Nevertheless, it is a subset of models which include such effects. As such, the interested modeler can, as a first step, use the technique outlined here for the simple material. In so doing, the modeler can gain further confidence in the validity of his numerical method that will be used for the real gas, for which there is no benchmark reactive flow solution.

Model Equations

The dimensionless conservation and constitutive equations, nomenclature, and scaling are taken from Ref. 1. The scaling is such that all postshock quantities are $\mathcal{O}(1)$. With the asymptotic method, the leading-order solution is an inert oblique shock attached to a wedge inclined at angle θ; the effects of heat release are accounted for at $\mathcal{O}(\epsilon)$. The equations are

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0 \tag{1}$$

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + \frac{1}{\rho}\frac{\partial P}{\partial x} = 0 \tag{2}$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + \frac{1}{\rho} \frac{\partial P}{\partial y} = 0 \tag{3}$$

$$u \frac{\partial P}{\partial x} + v \frac{\partial P}{\partial y} - \gamma \frac{P}{\rho} \left(u \frac{\partial \rho}{\partial x} + v \frac{\partial \rho}{\partial y} \right) = \epsilon(\gamma - 1) q \rho (1 - \lambda) e^{-\epsilon\Theta\rho/P} \tag{4}$$

$$u \frac{\partial \lambda}{\partial x} + v \frac{\partial \lambda}{\partial y} = (1 - \lambda) e^{-\epsilon\Theta\rho/P} \tag{5}$$

$$e = [1/(\gamma - 1)](P/\rho) - \epsilon\lambda q \tag{6}$$

$$P = \rho T \tag{7}$$

Dimensionless variables are the density ρ, velocities u and v, pressure P, product mass fraction λ, internal energy e, temperature T, and position coordinates x and y. Dimensionless parameters are the reciprocal of the square of the incoming Mach number ϵ, the heat release q, the ratio of specific heats γ, the activation energy Θ, and the wedge angle θ.

Initial preshock conditions are specified as $\rho = 1$, $u = \sqrt{\gamma}$, $v = 0$, $P = \epsilon$, and $\lambda = 0$. The inviscid solution must satisfy the downstream boundary condition of a zero-velocity component normal to the wedge surface. An additional outflow boundary condition must be specified for the numerical solution. Using an available code option, the outflow boundary was specified such that there were no gradients in flow variables. Such a condition is not present in the flow itself. The effects of this may be minimized by the fact that the flow at the outflow boundary is supersonic; hence, errors tend to propagate out of the computational domain. Solution of the asymptotic model requires the usual Rankine-Hugoniot relations for changes across the inert oblique shock discontinuity. Details are found in Ref. 1. For the numerical solution, specification of jump conditions is not required as a shock capturing scheme based on artificial viscosity is used to spread the discontinuity over a discrete number of cells.

Comparison Procedure

A series of oblique detonations was studied in which $7.5 \leq 1/\sqrt{\epsilon} = M_0 \leq 20$ and $0 \leq q \leq 10$. The other parameters were held constant at $\gamma = 7/5$, $\Theta = 0$, and $\theta = 20$ deg. The value of Θ is chosen to best compare with the asymptotic solution. The construction of the asymptotic solution formally requires that there be no reaction upstream of the shock. Downstream of the shock, the effects of the activation energy are not present in the hypersonic limit. It is possible to achieve this effect in numerical simulations by selecting any $\mathcal{O}(1)$ value of Θ. Alternatively, we have chosen to model this by introducing an ignition temperature such that there is no reaction before the shock and there is reaction after the shock. Such a feature is simply a convenience for this study and is not necessary or recommended for more general studies.

For each case, a numerical result and an asymptotic result were generated. All calculations were performed on a fixed grid of 19,701 (199 × 99) cell centers. The numerical method, which uses artificially large time steps to achieve rapid convergence of the unsteady equations to a steady solution, typically converged in 2000 pseudo time steps to a value in which the residuals were orders of magnitude less than the method's truncation error. The results presented here indicate that steady oblique detonation solutions to Eqs. (1–7) exist and that the numerical method used to predict them is convergent. The fact that the numerical method is not time-accurate prevents any conclusion to be drawn here regarding the hydrodynamic stability of the oblique detonation flowfield.

Next, the asymptotic method was used to calculate flow variables at each cell center. The method and solution are lengthy and described in detail in Ref. 1. Both solutions are then plotted for a qualitative comparison. For a quantitative comparison, the L_2 norm defined below gives the value of the average magnitude of the difference of the predictions of the two methods over the domain. Here the pressure differences are chosen, although any flow variable would do as well.

$$L_2 = \sum_{i=1}^{N} | P_{a_i} - P_{n_i} | / N \tag{8}$$

Here P_{a_i} and P_{n_i} are, respectively, the asymptotically and numerically predicted pressures at a given grid point, and N is the total number of grid points

Results

A case is presented in which there is good quantitative and qualitative agreement between the two methods, found when $M_0 = 20$ and $q = 10$. Figures 1a and 1b show asymptotic and numerical predictions of the pressure fields. The pressure contours have roughly the same topology, and the pressure magnitudes are close to one another. Small regions of disagreement are found near the shock, where the shock-capturing scheme spreads the pressure increase over a few cells, and near the outflow boundary, where a few small closed contours and small-amplitude oscillations in the contours are found. Similar agreement and disagreement are found when reaction progress and vorticity contours are compared.[3] In this and all cases, the domain is sufficiently large such that the flow has nearly completely relaxed to the equilibrium, irrotational state.

To better quantify the extent of the agreement, two values of L_2 were calculated and are presented in Figs. 2a and 2b. Figure 2a shows L_2 calculated along the wedge surface at points interior to the shock and downstream boundary. Along this line, L_2 has values consistent with the expected errors of the two methods, $\mathcal{O}(\epsilon^2)$ and $\mathcal{O}(\Delta x^2)$. To verify global agreement, L_2 was also calculated for the entire mesh shown in Fig. 2b. The trends are identical to the calculations over the wedge surface, but the values are an order of magnitude higher. This is likely due to the $\mathcal{O}(1)$ disagreement of predictions near the

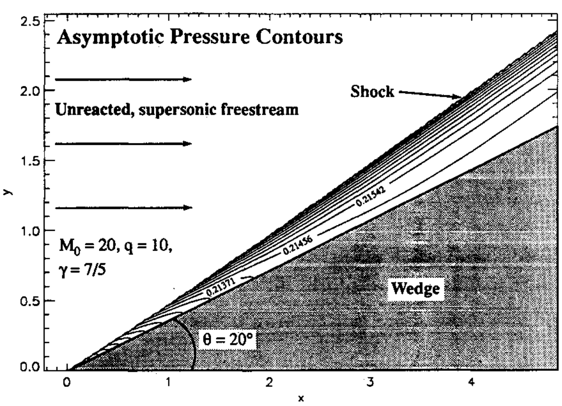

Fig. 1a Pressure contours for asymptotic solution ($M_0 = 20$, $q = 10$, $\gamma = 7/5$).

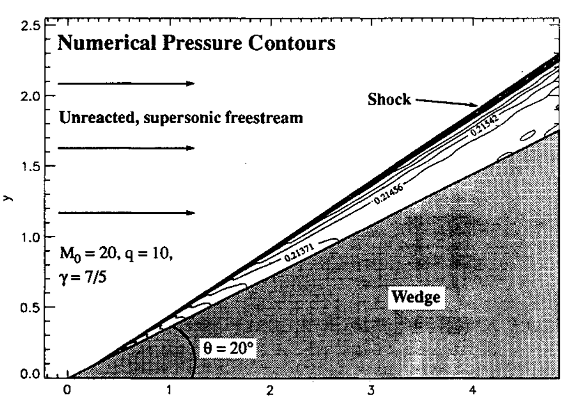

Fig. 1b Pressure contours for numerical solution ($M_0 = 20$, $q = 10$, $\gamma = 7/5$).

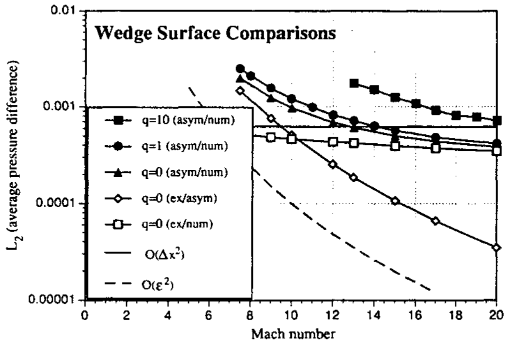

Fig. 2a Comparison along the wedge surface of exact, asymptotic, and numerical solutions using average pressure differences.

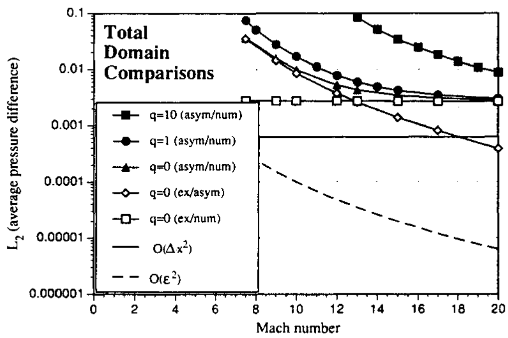

Fig. 2b Comparison over the entire domain of exact, asymptotic, and numerical solutions using average pressure differences.

shock due to inherent errors in the shock-capturing scheme and the error in the location of shock front predicted by the asymptotic method.

Calculations were first performed for $q = 0$ for $7.5 \leq M_0 \leq 20$. In this case the exact oblique shock solution is also available. From Fig. 2a, L_2 computed for the differences in the exact and numerical cases has a value near 4×10^{-4} which varies slowly with Mach number and is the same order of magnitude as the expected numerical error $[\mathcal{O}(\Delta x^2) = 6.25 \times 10^{-4}]$. L_2 was also computed for the differences between the asymptotic and exact solutions. As seen in Fig. 2a, this value of L_2 decreases with increasing Mach number, consistent with the asymptotic error estimate, which is $\mathcal{O}(\epsilon^2)$. L_2 is next calculated for the difference of the asymptotic and numerical methods with $q = 0$. At low supersonic Mach number, the difference is attributed to the error in the asymptotic method. For high Mach number, the difference is attributed to the error in the numerical method. A series of calculations was then performed for $q = 1$ and 10 over the same range of Mach numbers. For a given Mach number the value of L_2 increases with q. The curves follow the same trend as the $q = 0$ curve with increasing Mach number.

Final Remarks

Our results show that in addition to being useful as a means to gain basic understanding of high-speed reacting flows, asymptotic solutions of simple model problems are useful for making both qualitative and quantitative assessments of the numerical methods which are necessary for design of aerospace vehicles. By examining results which demonstrate the methods' basic agreement, such as those shown in Figs. 1a and 1b, additional confidence in the numerical method can be obtained. By studying results such as those shown in Figs. 2a and 2b, one can assess which approximation is responsible for the small differences that do exist. For example, the small differences in the results of Figs. 1a and 1b, obtained for $q = 10$, $M_0 = 20$, are primarily attributable to errors in the asymptotic method as seen in Figs. 2a and 2b. Nevertheless, the asymptotic solution shows that there is a nontrivial solution structure. Thus, one has a rational basis for distinguishing which structures predicted by the numerical method have a physical origin and which are numerical relics. As shown in detail in Ref. 3, as Mach number is held fixed and q is lowered, numerical errors overwhelm the effects of heat release so that the differences are primarily attributable to truncation errors in the numerical method.

In conclusion, it is recommended that the method of comparison given here be adopted as a new standard for numerical models of high-speed reacting flows.

Acknowledgments

This work was supported by the Jesse H. Jones Faculty Research Program and the NASA-ASEE Summer Faculty Fellowship Program. Thomas Jackson of Old Dominion University provided helpful suggestions for data presentation. The authors also thank Beverly Duncan and Shaye Yungster of NASA Lewis Research Center for assistance with the RPLUS code.

References

[1]Powers, J. M., and Stewart, D. S., "Approximate Solutions for Oblique Detonations in the Hypersonic Limit," *AIAA Journal,* Vol. 30, No. 3, 1992, pp. 726–736.

[2]Shuen, J.-S., and Yoon, S., "Numerical Study of Chemically Reacting Flows Using a Lower-Upper Symmetric Successive Overrelaxation Scheme," *AIAA Journal,* Vol. 27, No. 12, 1989, pp. 1752–1760.

[3]Powers, J. M., and Grismer, M. J., "Comparisons of Numerical and Exact Solutions for Oblique Detonations with Structure," AIAA Paper 91-1677, June 1991.

Supersonic Flutter of Composite Sandwich Panels

Le-Chung Shiau*
National Cheng Kung University, Tainan, Taiwan, Republic of China

Introduction

SANDWICH construction has been used in the aeronautical application for more than four decades since it offers the possibility of achieving high bending stiffness for small weight penalty. Today, there is renewed interest in using sandwich structures due to the introduction of new materials, such as advanced composite materials for the faces and nonmetallic honeycombs for core, which offer long awaited properties of both high stiffness and low specific weight. To use them efficiently a good understanding of their structural and dynamic behaviors under various loads is needed.

Panel flutter is a self-excited oscillation of the external skin of a flight vehicle and is caused by dynamic instability of inertia, elastic, and aerodynamic forces of the system. This type of aeroelastic instability has received much attention in the past 30 years.[1–4] As a result, this peculiar phenomenon is now reasonably understood for panels made of conventional isotropic materials. Recently, some works have been devoted to study the flutter characteristics of panels made of advanced composite materials,[5–8] but none of them deal with flutter of composite sandwich panels. In this Note, a flutter motion equation for a two-dimensional composite sandwich plate is derived by considering the total lateral displacement of the plate as the sum of the displacement due to bending of the

Received Nov. 21, 1991; revision received March 2, 1992; accepted for publication March 10, 1992.

*Professor, Institute of Aeronautics & Astronautics.