Numerical Solution of Two-Dimensional Turbulent Blunt Body Flows with an Impinging Shock

J. C. Tannehill* and Yvon C. Vigneron†
Iowa State University, Ames, Iowa
and
John V. Rakich‡
NASA Ames Research Center, Moffett Field, Calif.

Abstract

AN implicit finite-difference method has been developed to compute two-dimensional, turbulent, blunt-body flows with an impinging shock wave. The complete time-averaged Navier-Stokes equations are solved with algebraic eddy viscosity and turbulent Prandtl number models employed for shear stress and heat flux. The irregular-shaped bow shock is treated as a discontinuity across which the Rankine-Hugoniot equations are applied. A Type III turbulent shock interference flowfield has been computed and the numerical results compare favorably with existing experimental data.

Contents

The problem of shock interference heating arising from an extraneous shock wave impinging on a blunt body has been studied extensively during the past several years. In the Type III interference pattern, as described by Edney,[1] a shear layer originates at the intersection of the impinging shock and bow shock and interacts with the wall boundary layer. It has been shown[1,2] that the heat transfer is strongly dependent on whether the shear layer is laminar, transitional, or turbulent.

A method for computing laminar shock interference flowfields has been developed previously and applied to both two-dimensional[3,4] and three-dimensional[5] configurations. This method numerically computes the entire shock layer flowfield using the standard, unsplit, MacCormack finite-difference scheme[6] to solve the complete set of compressible Navier-Stokes equations. In order to calculate turbulent shock interference flowfields, a turbulence model was added to the method. Unfortunately, this explicit method suffers from the fact that the allowable time step is proportional to the square of the grid spacing in viscous regions.

In order to overcome this step size limitation, an implicit, noniterative, approximate-factorization, finite-difference scheme developed by Lindemuth and Killeen,[7] McDonald and Briley,[8] and Beam and Warming[9] is used in the present study to solve the complete time-averaged Navier-Stokes equations. Although this scheme increases the computation time per step over that of the previous explicit scheme by a factor of 1.8 for turbulent calculations, it permits a time step that is many times greater than the explicit time step when fine meshes are employed. Since turbulent shock interference flowfields have not been computed previously, relatively simple turbulence models are used in this initial study. In the turbulent boundary layer, the two-layer eddy viscosity model of Cebeci et al.[10] is used. In the turbulent shear layers, the following Prandtl mixing length model with an intermittency correction factor is employed:

$$\epsilon_2 = \rho\,(l/\delta_s)^2\delta_s^2(2S_{ij}S_{ij})^{1/2}/I_s \tag{1}$$

where δ_s is the shear layer width, (l/δ_s) the mixing length constant, S_{ij} the strain rate tensor, and I_s is the intermittency factor. In the interaction region, where the shear layer impinges on the boundary layer, a simple transition between the boundary-layer model and the shear-layer model is used. The modeling is illustrated in Fig. 1. Further details of the numerical procedure and governing equations are given in Ref. 11.

The implicit method was used to compute a Type III shock interference flowfield that corresponds to a two-dimensional turbulent experiment of Keyes.[2] The conditions of this test case were $M_\infty = 6.0$, $Re_{D_\infty} = 1.23\times10^6$, $Pr = 0.72$, $\gamma = 1.4$, $p_\infty = 1803.7$ N/m^2, $T_\infty = 60.5$ K, $p_{\rm stag} = 84{,}730$ N/m^2, $q_{\rm stag} = 37{,}451$ W/m^2, and $T_w = 408.3$ K. The impinging shock was inclined to the freestream at an angle of 22.67 deg and intersected the bow shock at $\theta = 14.4$ deg, where θ is the angle measured from the axis of symmetry of the body. A mesh consisting of 61 grid points normal to the body and 79 points around the body was used.

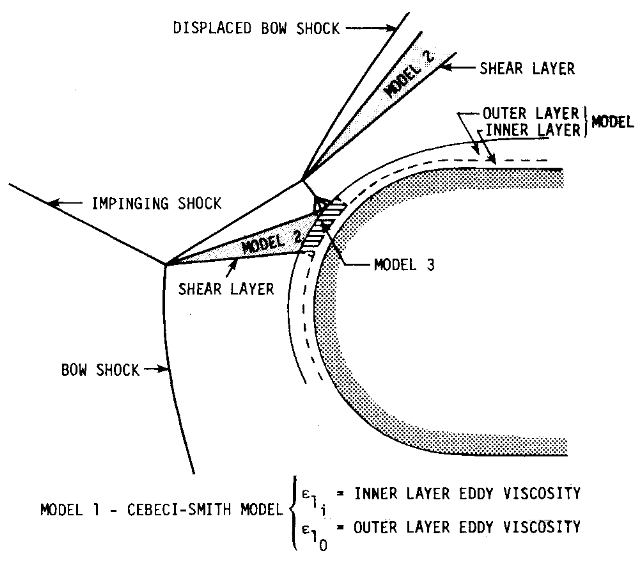

Fig. 1 Turbulence models for Type III interaction.

Presented as Paper 78-1209 at the AIAA 11th Fluid and Plasma Dynamics Conference, Seattle, Wash., July 10-12, 1978; submitted July 20, 1978, synoptic received April 30, 1979; revision received June 7, 1979. Full paper available from AIAA Library, 555 W. 57th St., New York, N.Y. 10019. Price: Microfiche, $2.00; hard copy, $5.00. **Order must be accompanied by remittance.**
Methods.

*Professor, Dept. of Aerospace Engineering and Engineering Research Institute. Associate Fellow AIAA.

†Research Assistant, Dept. of Aerospace Engineering; presently at SNIAS, Toulouse, France.

‡Research Scientist. Associate Fellow AIAA.

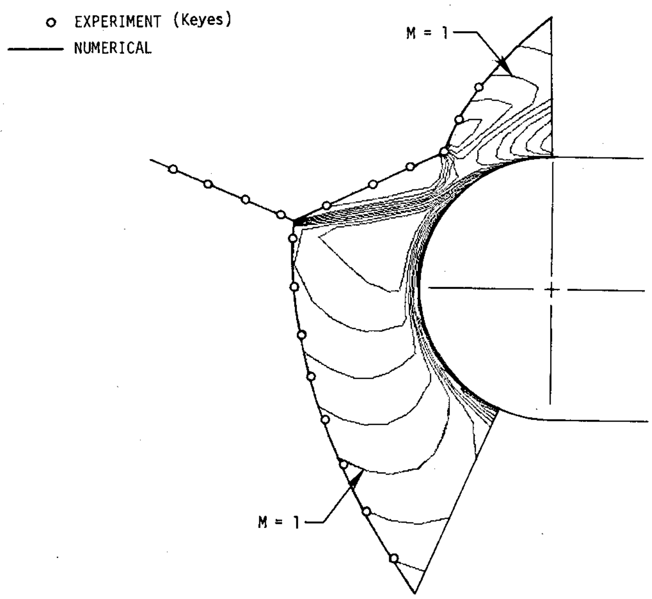

Fig. 2 Shock shape and Mach number contours.

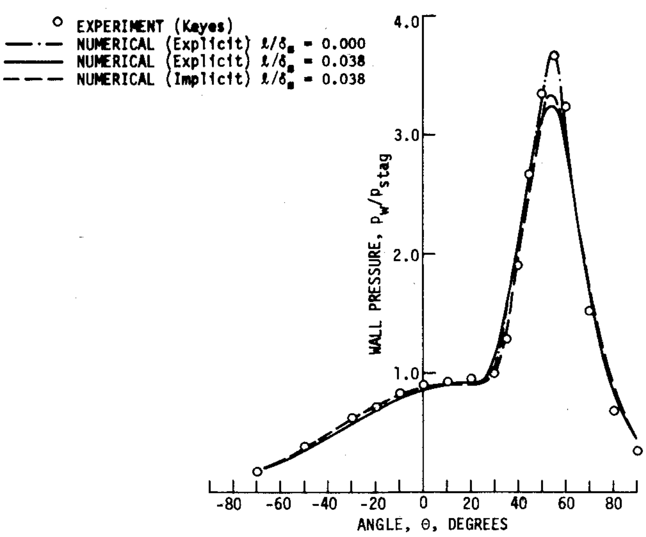

Fig. 3 Comparison of wall pressures ($\beta = 1.03$).

Numerical calculations were performed using two values of the grid clustering parameter β. The values were $\beta = 1.03$ and 1.002. Using the mildly refined mesh ($\beta = 1.03$), solutions were obtained with both the implicit method and the previous explicit method, which was modified to include the present turbulence modeling. The $\beta = 1.03$ mesh did not provide sufficient refinement near the wall to properly resolve the turbulent boundary layer for heat-transfer calculations, and a more highly refined mesh ($\beta = 1.002$) was used in the final implicit computation. The mixing length constant (l/δ_s) was taken to be 0.038 in all calculations, except for one explicit calculation where the shear layer was assumed laminar (i.e., $l/\delta_s = 0$).

The results of the calculations are shown in Figs. 2-4. The computed shock shape, which remained the same for all calculations, is compared with the experimental shock shape in Fig. 2. The agreement is quite good. The computed Mach number contours for the $\beta = 1.03$ implicit solution are also shown in Fig. 2 with increments (ΔM) of 0.2 starting at $M = 0$. A comparison between the numerical wall pressures and the experimental wall pressure is shown in Fig. 3. The explicit and implicit wall pressures computed with $l/\delta_s = 0.038$ agree reasonably well with each other. However, the best agreement with peak experimental pressure was achieved using $l/\delta_s = 0$. With $l/\delta_s = 0$, the shear layer is laminar and, therefore, more concentrated, which produces higher pressures and heat transfers at the wall. In the experiment, the shear layer was initially laminar and this may account for the better agreement with the $l/\delta_s = 0$ result. Heat-transfer results computed with the finer mesh ($\beta = 1.002$) are shown in Fig. 4. The computed peak heat-transfer rate is about 5.1 times the no-impingement stagnation point rate and this compares well with the experimental peak value of 5.0.

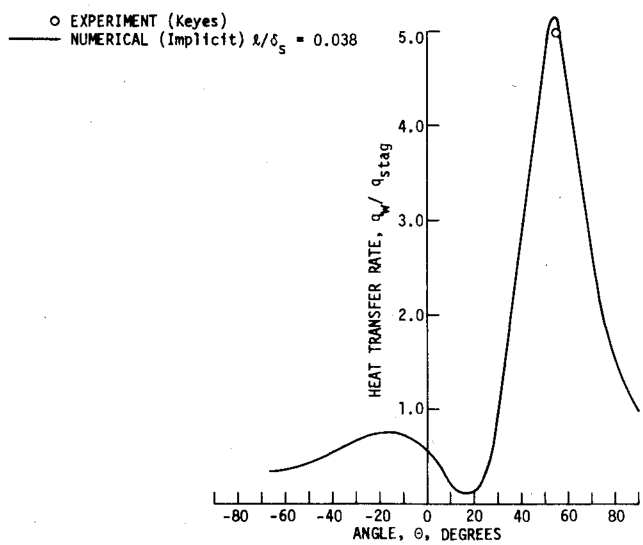

Fig. 4 Comparison of heat transfer ($\beta = 1.002$).

Acknowledgment

This work was supported by NASA Ames Research Center under Grant NGR 16-002-038 and by the Engineering Research Institute of Iowa State University.

References

[1] Edney, B. E., "Anomalous Heat Transfer and Pressure Distribution on Blunt Bodies at Hypersonic Speeds in the Presence of an Impinging Shock," FFA Rept. 115, The Aeronautical Research Institute of Sweden, Stockholm, Sweden, Feb. 1968.

[2] Keyes, J. W., "Correlation of Turbulent Shear Layer Attachment Peak Heating Near Mach 6," *AIAA Journal*, Vol. 15, Dec. 1977, pp. 1821-1823.

[3] Tannehill, J. C., Holst, T. L., and Rakich, J. V., "Numerical Computation of Two-Dimensional Viscous Blunt Body Flows with an Impinging Shock," *AIAA Journal*, Vol. 14, Feb. 1976, pp. 204-211.

[4] Tannehill, J. C., Holst, T. L., Rakich, J. V., and Keyes, J. W., "Comparison of a Two-Dimensional Shock Impingement Computation with Experiment," *AIAA Journal*, Vol. 14, April 1976, pp. 539-541.

[5] Holst, T. L., "Numerical Computation of Three-Dimensional Blunt Body Flow Fields with an Impinging Shock," Ph.D. Dissertation, Iowa State University, Ames, Iowa, 1975.

[6] MacCormack, R. W. and Baldwin, B. S., "A Numerical Method for Solving the Navier-Stokes Equations with Applications to Shock-Boundary Layer Interactions," AIAA Paper 75-1, Pasadena, Calif., Jan. 1975.

[7] Lindemuth, I. and Killeen, J., "Alternating Direction Implicit Techniques for Two-Dimensional Magnetohydrodynamic Calculations," *Journal of Computational Physics*, Vol. 13, Oct. 1973, pp. 181-208.

[8] McDonald, H. and Briley, W. R., "Three-Dimensional Supersonic Flow of a Viscous or Inviscid Flow," *Journal of Computational Physics*, Vol. 19, 1975, pp. 150-178.

[9] Beam, R. and Warming, R. F., "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations," *AIAA Journal*, Vol. 16, April 1978, pp. 393-402.

[10] Cebeci, T., Smith, A. M. O., and Mosinskis, G., "Calculation of Compressible Adiabatic Turbulent Boundary Layers," *AIAA Journal*, Vol. 8, Nov. 1970, pp. 1974-1982.

[11] Tannehill, J. C., Vigneron, Y. C., and Rakich, J. V., "Numerical Solution of Two-Dimensional Turbulent Blunt Body Flows with an Impinging Shock," AIAA Paper 78-1209, Seattle, Wash., July 10-12, 1978.