Characteristic-Based Flux Splitting to Solve Compressible Flow over an Airfoil

A. Atashbar Orang\textsuperscript{a,}*, S. E. Razavi\textsuperscript{b}

\textsuperscript{a}Mechanical Engineering Program, Middle East Technical University, Kalkanli, Turkey
\textsuperscript{b}Faculty of Mechanical Engineering, Tabriz University, Tabriz, Iran

1. Introduction

Computational Fluid Dynamics (CFD) has been found as the most important and effective tool in the engineering aspects especially with the rapid development of the computers in the recent decades; therefore, the numerical techniques are developing continuously, and new schemes and models are introduced to solve the flow field such as compressible flows around a NACA 0012 [1-3]. Some earlier studies have been done by solving potential flow for prediction of transonic flow with shock wave [4]. But, in the upper range of transonic regime through potential equations, results could not be trusted due to the ignoring the entropy variations and production of vortices [5]. On the other hand, Euler equations can predict well the transonic fluxes at the shocked regions [6]. As the first studies, Jameson et al. [1] solved the Euler Equations by finite volume method by adding artificial dissipation terms and Runge-Kutta scheme for time integration part. Thereinafter Roe proposed a scheme using method of characteristics based on the wave propagation of the flow information in the field, which could result more efficient solution of the Euler equations [7]. There is wide family of techniques to compute the fluxes, each with some specifications [8, 9]. Such studies have been focused on improvement of the numerical schemes regarding accuracy and cheaper solution. To achieve more accurate solution there are high-order methods with capability of shock capturing such as ENO and WENO schemes [10, 11]. As the matter of stability and faster convergence upwind schemes have been developed with better estimation of flux winding such as characteristics-based schemes [12]. To reduce computation effort there are also some techniques such as convergence accelerators like local time-stepping [13] by selection of the largest local time step based on the maximum possible CFL number. Residual smoothing is another accelerating technique by averaging of residuals to increase the value of the CFL number to speed up convergence. Other accelerating methods include adding enthalpy damping [14], using multi-grid [15], and mesh refinement [16]. Another aspect to have efficient numerical solution is the grid type used for the numerical computations. Orthogonal grid [17] and unstructured triangular grids perform well for complex domains [18].

* Corresponding author: orang@metu.edu.tr

This is an open access article under the CC-BY license (https://creativecommons.org/licenses/by/4.0/).

© Authors retain all copyrights.
In this study, the NACA 0012 airfoil as a benchmark in airfoil family is selected to analyze and predict the behavior of fluid around the airfoils in different conditions by the proposed CFS scheme. The goal of current study is to gain less expensive accurate numerical results in subsonic, transonic and supersonic regimes at different angles of attack. By finite volume method Euler equations are solved for the computational domain which is meshed by orthogonal grid with clustering near the wall. To test the efficacy of the proposed CFS scheme, convective terms are calculated and compared with the conventional JAM scheme. Characteristics-based boundary conditions are applied along with Runge–Kutta method for the time integration. The study leads improvement in accuracy, reduction of computation time; also, it can handle vortices at the downstream despite of absence of viscous effects.

2. Governing equations and Numerical approach

For inviscid compressible flow Euler equations is the governing equations of which two-dimensional conservative form in the Cartesian coordinates reads [19],

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial n} = 0,$$  \hspace{1cm} (1)

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} \rho u \\ \rho uu + p \cos \zeta \\ \rho vv + p \sin \zeta \\ \rho u_n + p \sin \zeta \end{bmatrix},$$  \hspace{1cm} (2)

where $\rho, u, v, u_n, p, E, H$ are density, velocity component is the $x$ direction, velocity component is the $y$ direction, normal component of velocity to the cell boundary, pressure, total energy, and total enthalpy. If we do surface integration of Eq. (1) for the generic control volume in Figure 1, we have

$$\frac{\partial}{\partial t} \int_{\Omega} \mathbf{U} dA + \int_{\partial \Omega} \mathbf{F}_n dL = 0,$$  \hspace{1cm} (3)

that in the discrete form one has

$$\frac{\partial \mathbf{U}}{\partial t} \Delta A + \sum_{k=1}^{4} \mathbf{F}_n^{(k)} \Delta L = 0,$$  \hspace{1cm} (4)

where superscript bar indicates the cell-averaged quantities. As shown in Figure 1, $\Delta A, \Delta L, u_\rho$ denote the cell area, cell boundary length, and parallel component of velocity to the cell boundary, respectively; one gets

$$\sin \zeta = \frac{\Delta x}{\Delta L}, \quad \cos \zeta = \frac{\Delta y}{\Delta L},$$

$$\Delta L = \sqrt{\Delta x^2 + \Delta y^2}, \quad \vec{N} = \cos \zeta \hat{i} + \sin \zeta \hat{j}, \quad u_x = u \cos \zeta, \quad u_y = v \sin \zeta, \quad u_\rho = \rho \cos \zeta - u \sin \zeta,$$  \hspace{1cm} (5)

Figure 1. Generic primary cell
2.1. Convective fluxes

2.1.1. Jameson Averaging Method (JAM)

Considering Figure 1, the flux at AB interface can be calculated by simple averaging as follows

\[
F_{\text{ab}} = 0.5 \left[ F_x(U_{i,j}) + F_x(U_{i+1,j}) \right].
\]

(6)

A similar procedure is followed for other sides.

2.1.2. Characteristics-based Flux Splitting (CFS)

The sort of hyperbolic system of equations in Eq. (1) can be written in characteristics form,

\[
\frac{\partial W}{\partial t} + \hat{A} \frac{\partial W}{\partial n} = 0,
\]

or

\[
\frac{\partial W}{\partial t} + \frac{\partial G}{\partial n} = 0,
\]

(7)

where

\[
W = \hat{L} U \hat{L} = \hat{R}^{-1},
\]

\[
G_x = \hat{A} W \Lambda_F.
\]

(8)

In Eq. (8) \(\hat{R}, \hat{A}, \text{ and } \hat{L}\) are averaged right eigenvector, eigenvalue, and left eigenvector matrices of Jacobian matrix \((A + \frac{\partial F_x}{\partial U})\), respectively. Each component of characteristic flux \(G_x\) can be splitted by Lax-Friedrichs flux splitting as

\[
g_x^\pm(w_j) = \frac{1}{2} (g_x(w_j) \pm \hat{a}_j w_j).
\]

(9)

Superscripts + and – denote the directions of upwinding in the Lax-Friedrichs flux splitting. Notice that

\[
\frac{d}{dw}g^+(w) \geq 0, \quad \frac{d}{dw}g^-(w) \leq 0.
\]

(10)

To define \(\hat{a}_j\) one can employ global flux splitting; here local splitting for the first-order accuracy is performed as

Following

\[
\hat{a}_j = \max \left| \hat{\lambda}_j(u_{i,j}) , \hat{\lambda}_j(u_{i+1,j}) \right|,
\]

(11)

where \(\hat{\lambda}_j\) denote the propagating speeds, which are eigenvalues of Jacobian matrix. Then the characteristic flux at AB interface is computed as

\[
G_{\text{ab}} = 0.5 \left[ G_x(U_{i,j}) + G_x(U_{i+1,j}) \right].
\]

(12)

Then we can convert back the characteristic fluxes to the conserved one by

\[
F_{\text{ab}} = \hat{R} G_{\text{ab}}.
\]

(13)

Also the introduced splitting method based on characteristics can be expressed by Godunov flux [20],

\[
F_{\text{ab}} = 0.5[ F_x(U_{i,j}) + F_x(U_{i+1,j}) ] + 0.5 \sum_{k=0}^{4} \hat{a}_k \hat{\delta}_{ik} \hat{\rho}^{(k)}.
\]

(14)

The corresponding eigenvalues and right-hand side eigenvectors read as
\[
\lambda_{(k)} = \begin{pmatrix}
    u_e - a \\
    u_e \\
    u_e + a
\end{pmatrix},
\quad \hat{\mathbf{R}} = \begin{pmatrix}
    1 & 0 & 1 \\
    \hat{u} - \hat{\alpha} \cos \zeta & \hat{\alpha} \cos \zeta & \hat{u} + \hat{\alpha} \cos \zeta \\
    \hat{v} - \hat{\alpha} \sin \zeta & \hat{\alpha} \sin \zeta & \hat{v} + \hat{\alpha} \sin \zeta \\
    \hat{H} - \hat{u}_e \hat{a} & \hat{u} \hat{a} & \hat{v}^2 + \hat{v} \hat{a} \hat{H} + \hat{u}_e \hat{a}
\end{pmatrix}
\]  

(15)

Wave's strength which is the difference of characteristic quantities between two cells, reads

\[
\hat{\delta}_{(k)} = \begin{pmatrix}
    \frac{\hat{\rho} \Delta u_e - \Delta p}{2a^2} \\
    \frac{\hat{\rho} \Delta u}{a} \\
    \frac{\Delta p}{a} - \frac{\Delta \rho}{a} \\
    \frac{\Delta p + \frac{1}{2} \hat{\rho} \Delta u}{2a}
\end{pmatrix}
\]

(16)

where \(a\) is sound speed and hatted quantities are the average values of two interacted cells. Roe introduced the following averaged values:

\[
\hat{\rho} = \frac{\rho_l \rho_r}{\rho}, \quad \hat{u} = \frac{\rho_l u_l + \rho_r u_r}{\rho_l + \rho_r}, \quad \hat{v} = \frac{\rho_l v_l + \rho_r v_r}{\rho_l + \rho_r}, \quad \hat{H} = \frac{\rho_l H_l + \rho_r H_r}{\rho_l + \rho_r},
\]

(17)

In CFS scheme averaging method is proposed based on local Mach number; for the first-order accuracy we have

\[
\xi_{(a)} = \frac{\text{Mach}_{i,j}}{\text{Mach}_{i,j} + \text{Mach}_{i+1,j}}, \quad \text{Mach}_{i,j} = \frac{V_{i,j}}{a_{i,j}}, \quad a_{i,j} = \sqrt{\frac{\rho_{i,j}}{\rho_{i,j}}}
\]

(18)

and for second-order accuracy

\[
\xi_{a+b} = \frac{\text{Mach}_{i,j}}{\text{Mach}_{i,j} + \text{Mach}_{i+1,j}}, \quad \xi_{a+b} = \frac{\text{Mach}_{i,j}}{\text{Mach}_{i,j} + \text{Mach}_{i+1,j}}, \quad \xi_{AB} = \frac{\xi_{a+b} + \xi_{a+b}}{2}
\]

(19)

and the hatted quantities at AB interface are defined as

\[
\hat{U}_{a+b} = \hat{\xi}_{a+b} \hat{U}_{i,j} + (1 - \hat{\xi}_{a+b}) \hat{U}_{i+1,j}.
\]

(20)

Since the Mach number has a significant role in propagation of information in compressible flows, applying those weighting factors improves the behavior of scheme regarding accuracy and convergence.

2.2. Artificial dissipation

To avoid numerical oscillations artificial dissipation is added to JAM scheme as follows

\[
\frac{d}{dt}(A \hat{U}_{i,j}) + Q(\hat{U}_{i,j}) - D(\hat{U}_{i,j}) = 0,
\]

(21)

where \(Q\) comprises the integrated convective terms. \(D\) is the artificial dissipation that can depreciate all adverse oscillations near the shock waves. A very efficient method to prescribe \(D\) is suggested as the combination of second-order and fourth-order differential term [21]:

\[
\hat{D}U = D(U + D(U),
\]

(22)
$$D_i U = d_{i+1/2, j} - d_{i-1/2, j}$$

$$D_j U = d_{i, j+1/2} - d_{i, j-1/2}$$

$$d_{i+1/2, j} = \frac{A}{\Delta x} \left[ \epsilon^{(2)}_{i+1/2, j} (U_{i+1, j} - U_{i, j}) - \epsilon^{(4)}_{i+1/2, j} (U_{i+2, j} - 3U_{i+1, j} + 3U_{i, j} - U_{i-1, j}) \right]$$

$$\epsilon^{(2)}$$ and $$\epsilon^{(4)}$$ coefficients are defined based on pressure

$$\nu_{ij} = \frac{p_{i+1, j} - 2p_{i, j} + p_{i-1, j}}{p_{i+1, j} + 2p_{i, j} + p_{i-1, j}}$$

$$\epsilon^{(2)}_{i+1/2, j} = k^{(2)} \max(\nu_{i+1, j}, \nu_{i, j})$$

$$\epsilon^{(4)}_{i+1/2, j} = \max \left( 0, k^{(4)} - \epsilon^{(2)}_{i+1/2, j} \right)$$

where fixed values can be used as $$k^{(2)} = \frac{1}{4}$$ and $$k^{(4)} = \frac{1}{256}$$. For the shocked region since $$\nu$$ is sensitive to the sharp pressure gradients, the value of $$\epsilon^{(2)}$$ is higher and consequently $$\epsilon^{(4)}$$ is equal to zero. On the other hand, in smooth area the value of $$\epsilon^{(2)}$$ is negligible and only the fourth-order dissipation is activated.

### 2.3. Boundary conditions

#### 2.3.1. Solid boundary

In the solid boundary the flow cannot penetrate which leads a physical boundary condition, which is imposed by means of eliminating the perpendicular component of velocity in the wall

$$q_{\perp b} = \overline{v_n} n = 0.$$ (27)

Considering locally on one-dimensional flow, we need two more boundary conditions which are obtained from the interior domain by two outgoing characteristics waves. The characteristics near wall can be written as

$$dc_1 = dp - \rho a \nu_n = 0,$$

$$dc_2 = dp - a^2 dp = 0,$$

$$dc_3 = dp + \rho a \nu_n = 0,$$ (28)

where $$dc_1$$ and $$dc_2$$ waves propagate towards solid boundary; then we can utilize them by discretization along those waves and we obtain

$$p_b - p_a - \rho a_y a_y (u_{ja} - u_{a, a}) = 0,$$

$$p_b - p_a - \rho a_y (\rho a_{y, a} - \rho_{b, a}) = 0,$$ (29)

where subscript $$b$$ indicates the values on the solid boundary and index $$ij$$ denotes the nearest cell to the boundary. Since $$u_{b, a} = 0$$, then combining Eq. 27, 28, and 29 gives the first order solid boundary conditions as

$$p_b = p_a - \rho a_y a_y u_{ja},$$

$$\rho_b = \rho_a + \frac{u_{ja, a}}{a_y} \rho_y.$$ (30)

#### 2.3.2. Far field boundary condition

Proper conditions for far field should minimize any spurious reflections. Again, we consider the locally one-dimensional flow along the perpendicular direction on the boundary. In the supersonic conditions all the characteristic waves are incoming and the far field values ought to be imposed for all the variables. For the subsonic case, there is one outgoing wave which must be extrapolated from interior domain. We can impose two physical conditions by setting the
density and pressure of the far filed \((\rho_{\infty}, p_{\infty})\) and find the normal velocity from outgoing wave \(d\eta_j\) in Eq. (28), extrapolated from interior domain as following [20]:

\[
p_{\eta_j} - p_{\infty} + \rho_{\eta_j}a_{\eta_j}(u_{\eta_j} - u_{b,n}) = 0,
\]

\[
u_{b,n} = u_{\eta,n} + \frac{p_{\eta_j} - p_{\infty}}{\rho_{\eta_j}a_{\eta_j}},
\]

where \(\eta_j\) denotes the nearest interior cell to the far field boundary.

### 3. Time integration and boundary conditions

The fifth-order Runge-Kutta method (RK5) is applied for time integration due to its wider stability range \((CFL \leq 3)\); RK5 method reads

\[
\frac{\partial U}{\partial t} + Q'(U) = 0, U^{(n+1)} = U^{(n)} - \epsilon_n \Delta t Q^{(n)}(U), \epsilon_n = 1/4, 1/6, 3/8, 1/2, 1, (m = 1, ..., 5),
\]

where \(Q^{*}\) is total residual including convective and dissipation terms. Variable time step is used which is dependent on the local flow changes and the grid spacing as,

\[
\Delta t = \frac{CFL \times \text{Max}(\Delta x_{ij} + \Delta y_{ij})}{\text{Max}[a + \sqrt{u^2 + v^2}], \text{Max}(\gamma_n)}
\]

\[
\gamma_n = \text{Max}[\{\rho_{\eta,n}, ||u_{\eta,n}||, ||v_{\eta,n}||, ||p_{\eta,n}||\}], \Delta x_{ij} + \Delta y_{ij} \times \gamma_n, n = 1, 2, 3, 4
\]

Error is computed as the largest value of difference between density, velocity and pressure at each time step which is normalized in the domain

\[
\text{Enorm} = \text{Max}\left((\rho^{(n+1)} - \rho^*), (\theta^{(n+1)} - \theta^*), (p^{(n+1)} - p^*)\right)
\]

### 4. Results and discussions

The proposed numerical framework is conducted to NACA 0012 airfoil in subsonic, transonic and supersonic regimes at angles of attack. Implementing CFS scheme allows increasing angle of attack even 90°. Figure 2 plots the grid which consists of 110×70 cells along radial and circumferential directions, respectively; for the outer boundary 15 chord lengths away from airfoil is set. Grid independence study is performed in transonic regime in zero angle of attack for three different resolutions by JAM scheme as reported in Figure 3. It is seen that there is negligible difference of Mach number distribution as mesh is refining, then 110×70 grid is taken as the acceptable mesh.

Figure 2. Grid for NACA 0012
For the convergence study a specific case with Mach=0.65 and AOA=2° is performed by CFS scheme and the result is compared with JAM scheme. Figure 4 presents the convergence history for this case where the comparison of convergence history between two schemes shows faster convergence for CFS scheme which is one of the advantages of proposed scheme to handle compressible flows. To validate performance of CFS scheme, the wall pressure distribution is plotted in Figure 5 and compared to that of AGARD report No.575 [22] at Mach=0.63 and AOA=2°. For both schemes CFL number is set to 1.

Contours of Pressure coefficient are given in Figure 5 at three different Mach numbers. Figure 6 illustrates the Mach number distribution at Mach=0.5, 0.6, 0.7 and AOA=0° where in all cases there is no sonic condition.

In Figure 7 streamlines for Mach=0.63 and AOA=16° are reported. Although there is no any factor to generate vortices in Euler equations such as viscosity, it is seen that the solution can capture the vortices. This can be explained by the induced numerical dissipation which leads to generate vortical structures at the wake region to satisfy the Kutta condition at the trailing age [23] or the baroclinical term of vorticity equation. Also, in Figure 8 vector plot at Mach=0.63 and AOA=16° is shown this fact.
Figure 5. Wall pressure coefficient distribution, NACA 0012, Mach=0.63, AOA=2°

Figure 6. Pressure coefficient contours, NACA 0012, CFS scheme

Figure 6. Wall Mach number distribution, NACA 0012, CFS scheme
Figure 7. Stream lines, Mach=0.63, NACA 0012, AOA=16°, CFS scheme

Figure 8. Vector plot, Mach=0.63, AOA=16°, NACA 0012, CFS scheme

Table 1 reported the lift coefficient at Mach=0.63 and different angles of attack where it is found that the stall happens nearly at AOA=17°.

| AOA° | 2   | 6   | 10  | 14  | 16  | 18  |
|------|-----|-----|-----|-----|-----|-----|
| C_L  | 0.32| 0.90| 1.23| 1.42| 1.46| 1.45|

Also by implementing CFS scheme transonic condition is studied at Mach=0.85 and AOA=1°. In Figure 9 wall pressure coefficient distribution is plotted where as it is seen, two shock wave occur at 0.85 and 0.6 of chord from leading edge in upper and lower surface, respectively.

Distribution of wall Mach number is reported in Figure 10 for the free stream Mach numbers of 0.8, 0.85 and 0.9 and at AOA=0° by CFS scheme. As it is expected, by increasing free stream Mach number the shock wave moves toward the trailing edge.
The proposed CFS scheme is also implemented to supersonic flow. In Figure 11 pressure contours for Mach=1.2 is plotted where the bow shock is observable in front of airfoil detached from leading edge. Also, to see the robustness of CFS scheme it is conducted to very high angle of attack even AOA=90° in this regimes as seen in Figure 11.
5. Conclusions

In present work, Characteristics-based Flux Splitting called CFS is proposed based on local Mach number to solve compressible flows over NACA 0012 in different sonic conditions and angles of attack. Time discretization, was done by fifth-order Runge-Kutta scheme. Local time stepping was utilized to accelerate convergence process. Consistent boundary conditions based on characteristics are adopted for solid and far field boundaries. CFS scheme computes convective flux more efficient in comparison with Jameson Averaging Method, JAM. Artificial dissipation is added to overcome spurious numerical oscillations in the shocked region. Due to the induced numerical dissipation vortices at the wake region in absence of viscous effects in smooth region leading to satisfy Kutta condition at trailing edge. CFS shows such robustness to handle flow at high Mach number and very high angles of attack even AOA=90°. The obtained results are compared with those of others and acceptable agreement is noticed in terms of accuracy.

References

[1] A. Jameson, W. Schmidt, E. Turkel. Numerical solution of the Euler equations by finite volume methods using Runge Kutta time stepping schemes. 14th fluid and plasma dynamics conference1981. p. 1259.

[2] H. Shan, L. Jiang, C. Liu. Direct numerical simulation of flow separation around a NACA 0012 airfoil. Computers & Fluids. 34 (2005) 1096-114.

[3] D.E. Raveh, A. Zaide. Numerical simulation and reduced-order modeling of airfoil gust response. AIAA journal. 44 (2006) 1826-34.

[4] A. Jameson. Numerical computation of transonic flows with shock waves. Symposium transsonicum II. Springer1976. pp. 384-414.

[5] A. Rizzi, H. Viviand. Numerical methods for the computation of inviscid transonic flows with shock waves. Notes on numerical fluid mechanics. 3 (1981) 3.

[6] J.P. Thomas, E.H. Dowell, K.C. Hall. Nonlinear inviscid aerodynamic effects on transonic divergence, flutter, and limit-cycle oscillations. AIAA journal. 40 (2002) 638-46.

[7] P.L. Roe. Characteristic-based schemes for the Euler equations. Annual review of fluid mechanics. 18 (1986) 337-65.

[8] B. Van Leer. Towards the ultimate conservative difference scheme. II. Monotonicity and conservation combined in a second-order scheme. Journal of computational physics. 14 (1974) 361-70.

[9] S.K. Godunov. Different Methods for Shock Waves Moscow State University, Russia, 1954.

[10] E. Weinan, C.-W. Shu. A numerical resolution study of high order essentially non-oscillatory schemes applied to incompressible flow. Journal of Computational Physics. 110 (1994) 39-46.

[11] G.-S. Jiang, C.-W. Shu. Efficient implementation of weighted ENO schemes. Journal of computational physics. 126 (1996) 202-28.

[12] A.A. Orang, A. Paykani. Modified characteristics-based schemes for compressible flow past an airfoil. Journal of Mechanics. 28 (2012) 627-35.

[13] A. Rizzi, L. ERIKSSON. Transfinite mesh generation and damped Euler equation algorithm for transonic flow around wing-body configurations. 5th Computational Fluid Dynamics Conference1981. p. 999.

[14] D.C. Jespersen. Enthalpy damping for the steady Euler equations. Applied numerical mathematics. 1 (1985) 417-32.

[15] Z. Wang. A fast nested multi-grid viscous flow solver for adaptive Cartesian/Quad grids. International Journal for Numerical Methods in Fluids. 33 (2000) 657-80.

[16] S. Nakamura, T. Holst. A new solution-adaptive grid generation method for transonic airfoil flow calculations. (1981).

[17] V. Akcelik, B. Jaramaz, O. Ghattas. Nearly orthogonal two-dimensional grid generation with aspect ratio control. Journal of Computational Physics. 171 (2001) 805-21.

[18] A. Jameson, D. Mavriplis. Finite volume solution of the two-dimensional Euler equations on a regular triangular mesh. AIAA journal. 24 (1986) 611-8.
[19] K.A. Hoffmann, S.T. Chiang. Computational fluid dynamics for engineers. Engineering Education System Wichita, KS1993.

[20] E.F. Toro. Riemann solvers and numerical methods for fluid dynamics: a practical introduction. Springer Science & Business Media2013.

[21] D. Goeken, O. Johnson. Fifth-order Runge-Kutta with higher order derivative approximations. Electronic Journal of Differential Equations. 2 (1999) 1-9.

[22] R. Lock. Test cases for numerical methods in two-dimensional transonic flows. ADVISORY GROUP FOR AEROSPACE RESEARCH AND DEVELOPMENT PARIS (FRANCE)1970.

[23] A. Kuethe, C. Chow. The finite wing. Foundations of Aerodynamics. John Wiley, New York, pp145-164. (1986).