JOURNAL OF AIRCRAFT
Vol. 41, No. 2, March–April 2004

Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Aerodynamic Analysis of Circular and Noncircular Bodies Using Computational and Semi-Empirical Methods

Shadi Mahjoob*
Aerospace Research Institute, 14665-834, Tehran, Iran
Mahmoud Mani†
Amir-Kabir University of Technology, Tehran, Iran
and
Mohammad Taeibi-Rahni‡
Sharif University of Technology, Tehran, Iran

Introduction

NOWADAYS, one of the open discussions in the area of missile aerodynamics is increasing the aerodynamic efficiency by optimizing the geometry of its cross section. In this regard, many researchers' attentions have been towards the missiles with noncircular cross sections. Even though this causes some inconvenience in the missile's stability and control, with new advances in that area the aerodynamists suggest noncircular cross sections without much concerns.

Jackson and Sawyer[1] have experimentally investigated bodies with elliptical cross sections and noticed a considerable increase in aerodynamic efficiency (lift-to-drag ratio) for horizontal elliptical cross sections (compared with circular cross sections). Also, the work of Graves,[2] which was performed for a wide range of Mach numbers (from subsonic to supersonic) around elliptic and circular cross sectional bodies, showed similar results. Recently, Sharma[3] investigated both computationally and experimentally a similar problem. His results not only support the previous findings, but also show increase in C_N and C_M (for horizontal elliptical cross sections). In addition, they indicate that, in any angle of attack, by increasing ellipticity ratio the aerodynamic efficiency is increased.

In families of missiles with noncircular cross-sections, because of the storage and carriage purposes the ones with square or rectangular cross sections (with round corners) are used more extensively, especially in tactical applications. The no–guidance air-to-ground missile, which needs to have the maximum possible load density and the minimum possible drag, is one of the typical missiles having noncircular cross sections.[1]

In 1987, Sigal and Lapidot[4] performed an extensive experimental investigation, in which three families of missiles with same length and cross-sectional areas were used. The shapes of their cross sections were square (with rounded corners), rectangular (with rounded corners), and circular. Their results showed that C_N was the highest for the horizontal rectangular case and the second highest for the square case. Of course, the problem with the rectangular case is that when the missile rolls and its cross section changes from horizontal rectangle to vertical its aerodynamic efficiency drops drastically (even to less than the circular case). Note, their results were consistent in both cases of a missile with and without fins.

Flow separation effects are highly related to the corners of the square or rectangular cross sections and cause unfavorable aerodynamic instabilities.[5] Many researchers have studied these corner effects, especially taking the radius of the round corners. These efforts have led to considerable decrease in the preceding unfavorable effects. Schneider[6] showed that for the square section the increase in the corner radius decreases the overall C_N and the roll angle effects.

Also, Daniel et al.[7] studied the preceding radius experimentally using a square cross section (with rounded corners) for different angles of attack. They found that the effects of this radius, especially in angles of attack higher than 10 deg, are considerable. Note, C_N also decreases as the corners become more rounded. According to their studies, in high angles of attack, for $r/b = 0.2$, C_N is close to the circular cross-section case ($r/b = 0.5$), whereas for $r/b = 0.1$, C_N is close to the square cross-section case ($r/b = 0.0$).

This test has also been performed for a case in which four fins are attached to the end of the missile. For this case, similar results have been obtained. Of course, because the fins are the main devices for producing lift they cause reduction in the effects of the corner radii in the value of C_N for compound body-fin configurations. However, to the same degree the corner radii affect C_N.

In our work, we have compared the aerodynamic characteristics of two body having the same cross-sectional areas, but with different shapes (one circular and one square with round corners) in a transonic regime at different angles of attack. To differentiate the noncircularity and the fin effects, we have considered the bodies with no fins. This study has been done using a computational-fluid-dynamics (CFD) code and two semi-empirical codes not only to obtain aerodynamic parameters C_L and C_D, but also to study the physics of flow.

Computational Methodologies

The FLUENT CFD code,[8] which uses a cell-centered finite volume method and has been proven to work well for different flow regimes around missiles,[9,10] was used in this study. The implicit method implemented uses a segregated solution method. Note, all of the schemes used here are second order. The SIMPLE algorithm with underrelaxation coefficients is used in the overall discretization of the equations. To reduce the dispersion errors (and also to increase the speed of the computations), the multigrid approach has also been used.

Physical Properties

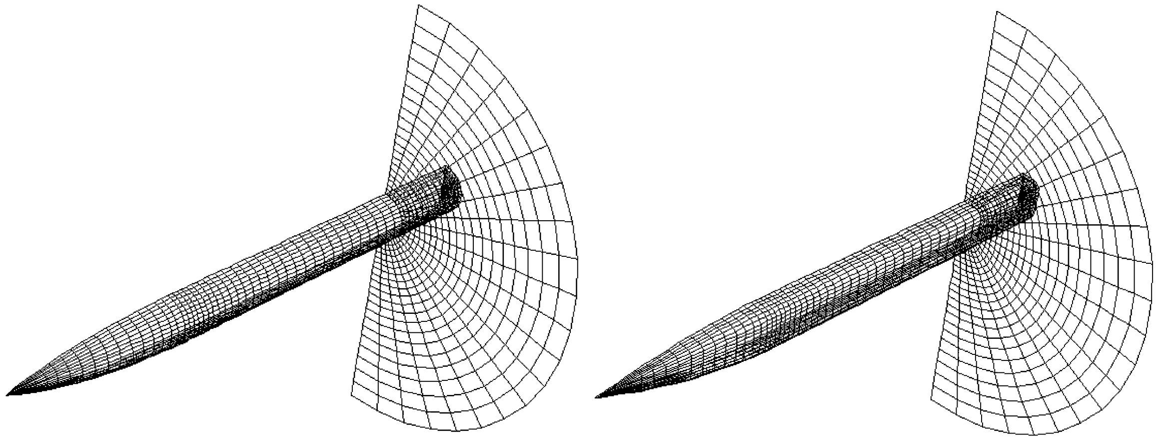
The flow considered here is three-dimensional, compressible, stationary, single-phase, viscous, and turbulent so in which the standard

Presented as Paper 2002-2941 at the AIAA 20th Applied Aerodynamics Conference, St. Louis, Missouri, US 25 June 2002; received 16 February 2003; revision received 7 August 2003; accepted for publication 25 November 2003.

*Research Engineer, Hava-Faza Alley, Mahestan St., Iran-Zamin St., Shahrak-Gharb; shmahjoob@ari.ac.ir. This work is resulted from her M.Sc. thesis in Amir-Kabir University of Technology.

†Associate Professor, Aerospace Engineering Department, Somaie St., Hafez Avenue; mani@cic.aut.ac.ir. Senior Member AIAA.

‡Assistant Professor, Aerospace Engineering Department, Azadi Avenue; taeibi@sharif.edu.



a) b)

Fig. 1 Two body fuselages and computational grid: a) the circular body with one plane at an axial location and b) the square body with one plane at an axial location.

κ–ε model[11] has been used for turbulence modeling. The fluid is air, for which the viscosity is obtained using Sutherland relation. The freestream Mach number is 0.8, the flow Reynolds number based on body length is 1.83×10^6, the static temperature is 268.305K, the total gauge pressure is 29066.31 Pa, the turbulence intensity is 10%, the characteristic length scale (the body diameter of cross section) is 0.04 m, and the operating pressure is 55433.69 Pa.

Geometric Modeling

In this study, the aerodynamics of the bodies of two body, one having a square and the other having a circular cross section (with same cross-sectional areas), have been studied (Fig. 1). Note, because of the symmetric assumption used only half of the domain is considered here. Our computational results indicate that this assumption has not affected the results considerably. Also note that the outer boundaries of the computational domain have been assumed to be nine radii in the radial direction, two body lengths at the front, and three body lengths behind the body. This size of the domain was shown to be optimal using numerical experimentation. A structured, body fitted, and nonuniform grid has been used in this study. The grid independency was studied and as a result a $169 \times 21 \times 21$ grid was chosen.

Boundary Conditions

Different boundaries of the physical domain have been used in this work. The wall of the body is assumed to be adiabatic. The freestream pressure is assumed at the inlet and outlet of the domain. In the pressure outlet boundary condition, only the static pressure is specified, and all other flow quantities are extrapolated from the flow in the interior. The far-field pressure condition is also used. This boundary condition is often called a characteristic boundary condition because it uses characteristic information (Reiman invariants) to determine the flow variables at the boundaries. Finally, at the symmetric plane the symmetric boundary condition has been implemented.

Discussion of Results

To investigate the effects of the shape of the cross sections, two different cases were studied in this work. The computations were performed for different angles of attack, namely, 0.0, 2.06, 4.09, 8.18, and 16.27 deg. The results have been compared with the existing benchmark data and with the data we obtained using two different semi-empirical codes.

The results are mainly in the form of aerodynamic coefficients. The benchmark data used here are the results obtained by Mani and Khajehfard.[12] Note, the only difference between their and our body is that they also have two thin fins connected to the end of their body. To investigate solely the body's cross-sectional shape effects (without the aerodynamic influences of the fins), as mentioned in the Introduction, we considered the body-alone configuration. Because the existing wind-tunnel results include fins, it is not possible

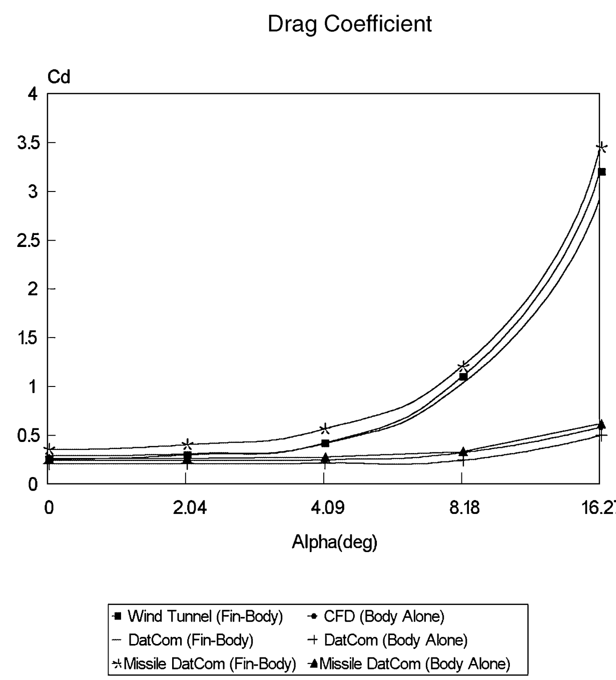


Fig. 2 Comparison of the drag coefficients for fin-body combination and body-alone using wind tunnel, CFD code, digital DatCom, and missile DatCom.

to perform a direct comparison between our computational and the wind-tunnel results (especially at nonzero angles of attack). This is why, for comparison purposes, we also calculated the aerodynamic characteristics using digital DatCom[13] and missile DatCom[14] softwares. These two semi-empirical codes can calculate static stability, high lift and control, and dynamic derivative characteristics of airplanes and missiles. Note, the reference area throughout this work is assumed to be the body cross-sectional area.

First, using the digital DatCom software, we obtained the aerodynamic coefficients for the body used by Mani and Khajehfard.[12] Figures 2 and 3 have compared drag coefficient and lift coefficient for the circular cross-section case. In general, digital DatCom results can have up to about 20% error. Also, there are errors in the preceding experimental measurements because of the lack of a dryer in their wind tunnel, the model's manufacturing errors, and the tunnel being very old.[12] Therefore, the 7.2% error in drag coefficient and 18% error in lift coefficient of Figs. 2 and 3 are acceptable, and thus our digital DatCom results can be safely used for the verification of our CFD results. Actually, the preceding calculations were also performed using missile DatCom software (Figs. 2 and 3). The errors were 21% in drag coefficient and 9.5% in lift coefficient.

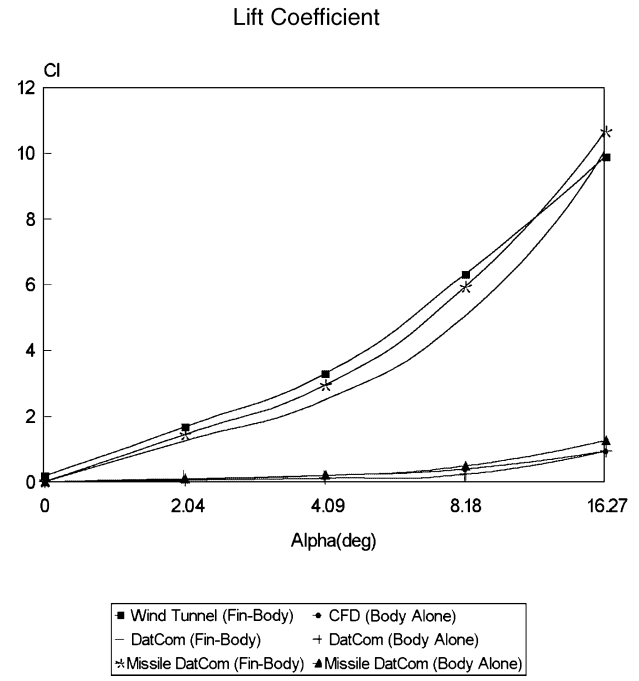


Fig. 3 Comparison of the lift coefficients for fin-body combination and body-alone using wind tunnel, CFD code, digital DatCom, and missile DatCom.

Figures 2 and 3 also compare the CFD results and the results obtained using digital DatCom software for the body-alone used in this study. For this case, the errors for C_D and C_L were 10.5 and 25%, respectively. However, using missile DatCom, the related errors were 16 and 11.3% for C_D and C_L, respectively. Our CFD results, which have about the same amount of error as the wind-tunnel data, were somewhere between the digital DatCom and the missile DatCom results, which can reassure our CFD results. The error percentages also show that in both bodies with and without fins digital DatCom generates less error for C_D and missile DatCom makes less error for C_L. The study of Figs. 2 and 3 also show the drastic effects of the fins on the aerodynamic coefficients.

Because the fins used in the just-mentioned experimental work are thin and symmetric, at zero angle of attack the corresponding results can directly be used for our CFD verification purposes. At zero angle of attack and $M = 0.8$, the fins' drag is mainly caused by friction. This is because, at this Mach number, the wave drag is relatively small. Also, because we have zero lift coefficient the related induced drag coefficient ($\sim C_L^2/\pi e AR$) is zero as well. Finally, since the fins are thin, their pressure drag is also negligibly small. Thus, at zero angle of attack the only remaining drag is in the form of skin friction. Note that the friction drag of the body-alone configuration is available from our CFD results. Now, because the surface area of the body and the fins are known we can estimate the friction drag of the fins. Here, the interference drag has been neglected, and the authors believe it is very small for the conditions used. If we subtract the friction drag of the fins (which is almost their total drag) from the wind-tunnel result (the overall drag of body-fin configuration), we can have the total drag of the body-alone configuration, which can be used as benchmark data for our CFD computations. With regard to the assumptions used, this comparison shows satisfactory agreements. Note the zero-lift result, obtained from our CFD computations, matches the wind-tunnel results, as expected.

Figure 4 shows better aerodynamic performance of the squared section body (compared with the circular one) at different angles of attack. This is because we have less drag and more lift in the square case. As the angle of attack is increased, the lift and drag coefficients increase as well as expected.

Another point is that the friction drag is about 30% of the overall drag in both circular and square cases. As expected, this drag is more for the square case because of its larger surface area. However, note

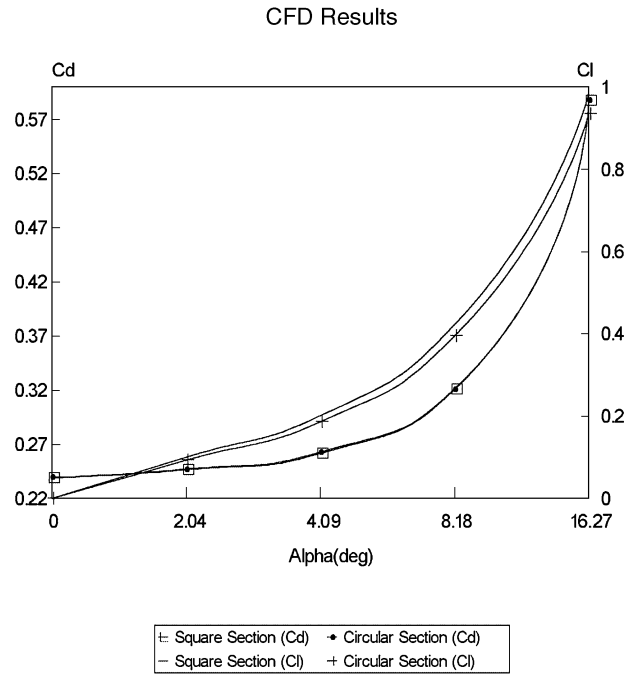


Fig. 4 Comparison of drag and lift coefficients for square and circular bodies (CFD results).

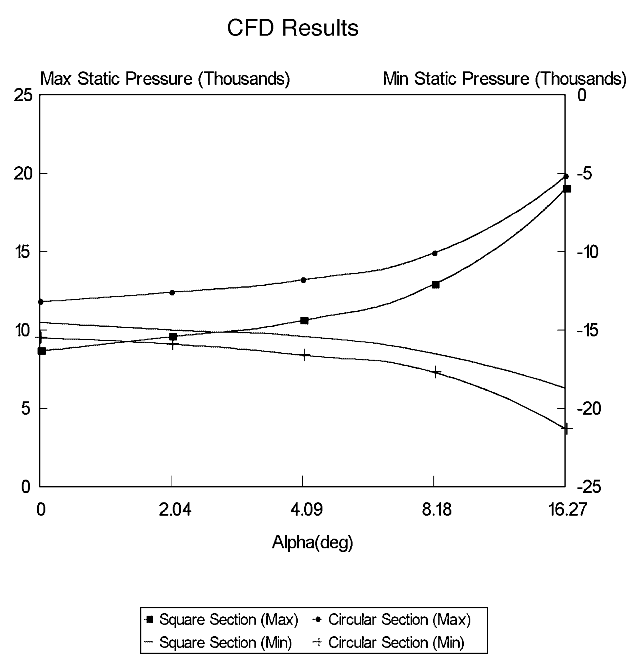


Fig. 5 Comparison of maximum and minimum static pressure for square and circular bodies (CFD results).

that the overall drag of the square case is somehow smaller than the circular case.

The study of flow physics shows that the pressure difference between the front and the back of the body that produce pressure drag is more in the circular body. Figure 5 shows maximum and minimum pressures for the two circular and square body at different angles of attack. From this figure, note that the circular body has higher maximum and lower minimum at all different angles of attack. In other words, the circular body has higher pressure differences at all those angles of attack.

In Fig. 6, the aerodynamic results obtained from both semi-empirical codes for body-alone configuration are shown. Regarding the mentioned error percentages of these codes to calculate C_L and C_D, we have used the results of digital DatCom for studying C_D and missile DatCom for C_L. Similar to CFD results, this figure indicates higher aerodynamic efficiency for square body.

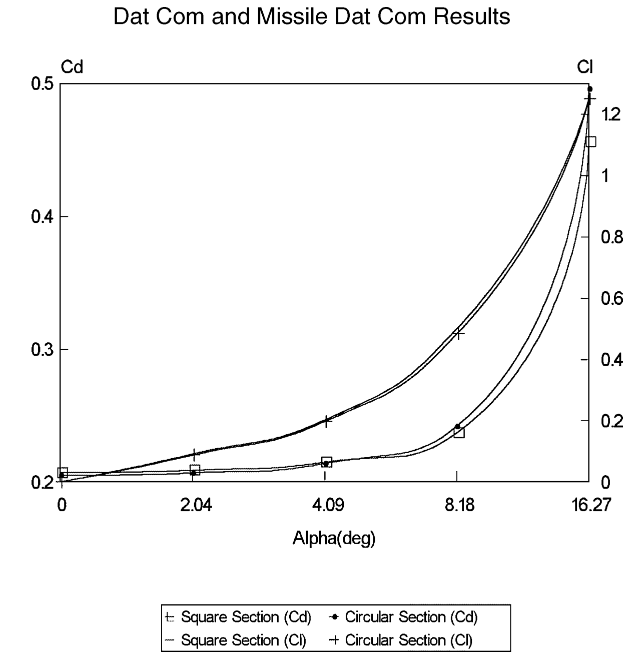


Fig. 6 Comparison of drag and lift coefficients for square and circular bodies (semi-empirical results).

Conclusions

In order to investigate the effects of the shape of the cross-sections, two different cases, circular and square with round corners, were studied in this work using computational and semi-empirical methods. The results that are mainly in the form of aerodynamic coefficients show better aerodynamic performance for the square section missile at different angles of attack. Because the square body, in comparison with the circular one, causes more lift and a bit less drag. It is noticeable that although the friction drag is more for the square case due to its larger surface area, the overall drag of the square case is somehow smaller than that of the circular case.

References

[1]Jackson, C. M., and Sawyer, W. C., "Bodies with Non-Circular Cross-Sections and Bank-to-Turn Missiles," *Tactical Missile Aerodynamics: General Topics*, edited by M. J. Hemsch, Vol. 141, Progress in Astronautics and Aeronautics, AIAA, Washington, DC, 1991, pp. 365–389.

[2]Graves, E. B., "Aerodynamic Characteristics of a Mono-Planar Missile Concept with Bodies of Circular and Elliptical Cross-Sections," NASA TM-74079, Dec. 1977.

[3]Sharma, R. K., "Experimental Aerodynamic Characteristics of Elliptical Bodies with Variation in Ellipticity Ratio," *Proceedings of the 18th AIAA Applied Aerodynamics Conference and Exhibit*, Denvor, CO, USA, Aug. 2000.

[4]Sigal, A., and Lapidot, E., "The Aerodynamic Characteristics of Configurations Having Bodies with Square, Rectangular, and Circular Cross-Sections at a Mach Number of 0.75," *AIAA Atmospheric Flight Mechanics Conference*, Monterey, CA, Aug. 1987.

[5]Nielsen, J. N., "Problems Associated with the Aerodynamic Design of Missile Shapes," *Proceedings of the Second Symposium on Numerical and Physical Aspects of Aerodynamic Flows*, edited by T. Cebeci, California State Univ., Springer, CA, USA, Jan. 1983.

[6]Schneider, W., "Experimental Investigation of Bodies with Non-Circular Cross-Section in Compressible Flow," *Symposium on Missile Aerodynamics*, AGARD-CP-336, Trondheim, Norway, 20–22 Sept. 1982, pp. 19-1–19-15.

[7]Daniel, D. C., Yechout, T. R., and Zollars, G. J., "Experimental Aerodynamic Characteristics of Missiles with Square Cross-Sections," *Journal of Spacecraft and Rockets*, Vol. 19, No. 2, 1982.

[8]"FLUENT 5 User's Guide," FLUENT Inc., Lebanon, New Jersey, July 1998.

[9]Birch, T. J., Wrisdale, I. E., and Prince, S. A., "CFD Predictions of Missile Flow-Fields," *Proceedings of the 18th AIAA Applied Aerodynamics Conference and Exhibit*, Denvor, CO, USA, Aug. 2000.

[10]Bulbeck, C. J., Morgan, J., and Fairlie, B. D., "RANS Computations of High-Incidence Missile Flow Using Hybrid Meshes," *Proceedings of the 18th AIAA Applied Aerodynamics Conference and Exhibit*, Denvor, CO, USA, Aug. 2000.

[11]Keimasi, M. R., and Taeibi-Rahni, M., "Numerical Simulation of Jets in a Cross-Flow Using Different Turbulence Models," *AIAA Journal*, Vol. 39, No. 12, 2001, pp. 2268–2277.

[12]Mani, M., and Khajehfard, A., "Aerodynamic Characteristics of Bodies with Square, Elliptic and Circular Cross-Sections," *Proceedings of the 18th AIAA Applied Aerodynamics Conference and Exhibit*, Aug. 2000.

[13]"The USAF Stability and Control Digital DatCom," The Air Force Wright Aeronautical Lab., AFWAL,/FIGC, McDonnell Douglas Astronautics Co., OH, April 1979.

[14]Vukelich, S., R., Stoy, S. L., and Moore, M. E., "The USAF Automated Missile DatCom," AFWAL-TR-86-3091, McDonnell Douglas Missile Systems Co., 1986.

Incorrectness of the *k* Method for Flutter Calculations

Mayuresh J. Patil*
Virginia Polytechnic Institute and State University, Blacksburg, Virginia 24061
Rudolph N. Yurkovich†
The Boeing Company, St. Louis, Missouri 63166
and
Dewey H. Hodges‡
Georgia Institute of Technology, Atlanta, Georgia 30332-0150

Introduction

IT has been appreciated by aeroelasticians for many years that results from the V-g or k method of flutter analysis can be difficult to interpret or even misleading.[1–3] The purpose of the present Note is to consider this difficulty with the k method in an especially simple setting, that is, with steady-flow aerodynamics. As Pines[4] showed many years ago in a different context, the use of highly simplified aerodynamics can be especially illuminating. In the special case of steady-flow aerodynamics and zero structural damping, the principal result of the present Note shows one way that the V-g or k method can lead to difficulties. For this special case, a particularly clear understanding of the nature of these difficulties can be obtained. This Note adds a modest but new and, the authors believe, helpful addition to the rich literature on this perplexing issue.

It is well known that k and p methods of calculating flutter speeds are quite different.[5] It is frequently assumed that both methods calculate the same flutter point because at zero damping (sinusoidal motion) both the methods are mathematically the same.[6] Nevertheless, the authors have come across cases in which the flutter speed, as defined by the speed after which the damping measure is negative (i.e., the real part of the eigenvalue is positive in p method, and artificial damping is positive in k method) or the speed at which frequency coalescence occurs, is predicted differently by the two methods. The inconsistency of the k method in predicting the flutter speed can be directly observed in systems without viscous damping (i.e., without forces proportional to velocity) in either the structural or aerodynamic operators. Such a system is neutrally stable for all

Received 26 March 2003; revision received 9 September 2003; accepted for publication 25 November 2003.

*Assistant Professor, Department of Aerospace and Ocean Engineering. Member AIAA.

†Boeing Technical Fellow. Fellow AIAA.

‡Professor, School of Aerospace Engineering. Fellow AIAA.