The Effects of Rotational Flow, Viscosity, Thickness, and Shape on Transonic Flutter Dip Phenomena

T.S.R. Reddy
The University of Toledo
Toledo, Ohio

Rakesh Srivastava
Georgia Institute of Technology
Atlanta, Georgia

and

Krishna Rao V. Kaza
Lewis Research Center
Cleveland, Ohio

Prepared for the
29th Structures, Structural Dynamics and Materials Conference
cosponsored by the AIAA, ASME, ASCE, and AHS
Williamsburg, Virginia, April 18–20, 1988
THE EFFECTS OF ROTATIONAL FLOW, VISCOSITY, THICKNESS, AND SHAPE ON TRANSONIC FLUTTER DIP PHENOMENA

T.S.R. Reddy*  
The University of Toledo  
Toledo, Ohio 43606  

Rakesh Srivastava  
Georgia Institute of Technology  
Atlanta, Georgia 30332  

and  

Krishna Rao V. Kaza  
National Aeronautics and Space Administration  
Lewis Research Center  
Cleveland, Ohio 44135

Abstract

The transonic flutter dip phenomena on thin airfoils, which are employed for propfan blades, is investigated using an integrated Euler/Navier-Stokes code and a two degrees of freedom typical section structural model. As a part of the code validation, the flutter characteristics of the NACA 64A010 airfoil are also investigated. In addition, the effects of artificial dissipation models, rotational flow, initial conditions, mean angle of attack, viscosity, airfoil thickness, and shape on flutter are investigated.

The present results obtained with a Euler code for the NACA 64A010 airfoil are in reasonable agreement with published results obtained by using transonic small disturbance and Euler codes. The two artificial dissipation models, one based on the local pressure gradient scaled by a common factor and the other based on the local pressure gradient scaled by a spectral radius, predicted the same flutter speeds except in the recovery region for the case studied.

The effects of rotational flow, initial conditions, mean angle of attack, and viscosity for the Reynold's number studied seem to be negligible or small on the minima of the flutter dip. However, they have significant effect on the flutter boundary away from the dip.

The flutter dip shifts towards higher Mach number as the thickness decreases for symmetrical airfoils, other parameters being the same. This is in direct relation to the location and strength of the shock. The flutter boundary for a thin cambered airfoil (propfan airfoil) showed that the effect of camber is nullified by the effect due to reduction in thickness and showed a relatively low transonic dip.

The flutter boundary of a simulated SRS propfan typical section model showed a very low transonic flutter dip. However, further studies with varying mean angles of attack and mass ratio are required to better understand the transonic flutter dip phenomena of highly swept propfans.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>a_h</td>
<td>distance of elastic axis from the origin</td>
</tr>
<tr>
<td>a_m</td>
<td>speed of sound</td>
</tr>
<tr>
<td>b</td>
<td>semichord</td>
</tr>
<tr>
<td>C_h</td>
<td>damping coefficient in plunging</td>
</tr>
<tr>
<td>C_p</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>C_α</td>
<td>damping coefficient in pitching</td>
</tr>
<tr>
<td>c</td>
<td>chord</td>
</tr>
<tr>
<td>c_l</td>
<td>lift coefficient</td>
</tr>
<tr>
<td>c_m</td>
<td>moment coefficient about elastic axis</td>
</tr>
<tr>
<td>e</td>
<td>total energy of the fluid per unit volume</td>
</tr>
<tr>
<td>h</td>
<td>plunging (bending) displacement, positive downward</td>
</tr>
<tr>
<td>I_α</td>
<td>polar mass moment of inertia</td>
</tr>
<tr>
<td>J</td>
<td>Jacobian of transformation</td>
</tr>
<tr>
<td>K_h</td>
<td>plunging spring constant</td>
</tr>
<tr>
<td>K_α</td>
<td>pitching spring constant</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
</tr>
<tr>
<td>m</td>
<td>mass per unit length</td>
</tr>
<tr>
<td>Q_h</td>
<td>generalized force in plunging</td>
</tr>
<tr>
<td>Q_α</td>
<td>generalized force in pitching</td>
</tr>
<tr>
<td>R_e</td>
<td>Reynold's number based on chord</td>
</tr>
<tr>
<td>r_α</td>
<td>radius of gyration about elastic axis</td>
</tr>
<tr>
<td>r_c_g</td>
<td>radius of gyration about center of gravity</td>
</tr>
<tr>
<td>S</td>
<td>static unbalance, mbx_α</td>
</tr>
</tbody>
</table>

*NASA Resident Research Associate.
I models for three-dimensional subsonic flow condi-
I phs of the flutter phenomena and to validate
experiment varied from poor to good, depending on
the recently developed linear cascade aeroelastic
test parameters.

The requirements of high aerodynamic effi-
ciency and low noise in the operating range of
transonic flow has resulted in thin (2 to 4 percent
thick), highly swept and twisted blades for prop-
fans. These blades operate at moderate to large
blade sweep. The correlation between theory and
cascade aerodynamic theory with a correction for
viscosity on the transonic flutter characteristics
of a thick airfoil are also investigated.

The approach being used for the flutter anal-
ysis is the simultaneous integration of structure-
fluid dynamics equations. The loads determined
by solving the fluid dynamic equations are used
as input to the structural dynamic equations. The
resultant deflections are then used as input to the
fluid dynamic equations to determine the loads
again. If the amplitude of the deflections grow in
time, then flutter is said to have occurred. In
all methods of flutter analyses, the difference
lies mainly in the prediction of aerodynamic loads.

Early published research on the effects of
transonic flow on propfan blade, or propfan airfoil
flutter characteristics is nonexistent. However,
considerable research7-24 is being conducted on
flutter characteristics of airfoils and aircraft
wings in transonic flow. Published results20-24
show that there is a considerable amount of reduc-
ition in flutter speed in the transonic flow regime
for highly swept blades when compared to the flut-
er speed predicted by linear theories. This
reduction has been called transonic flutter dip.
This transonic flutter dip has been investigated
by using transonic small disturbance (TSD)
th理论:10,20,21 modified strip analysis method:22
the Euler equations:23 and the Navier-Stokes equa-
tions.24 Many of these methods, along with flutter
results, are reviewed in Ref. 25.

The small disturbance theory and full poten-
tial flow theories fail to take into account the
rotationality of the flow, which could be signifi-
cant for regions where curved shocks exist. Fur-
thermore, potential flow theory may give multiple
steady-state solutions at a given flight condition
if the free stream Mach number is sufficiently

\[ t \] time
\[ \xi, \eta \] nondimensional time, \( t a_0 / c \)
\[ u, v \] Cartesian velocities normalized by the
speed of sound
\[ V \] resultant velocity
\[ V_F \] flutter velocity
\[ \bar{x} \] distance measured from leading edge to
trailing edge, normalized by chord
\[ x, y \] Cartesian coordinate system
\[ x_\alpha \] distance between elastic axis and cen-
ter of gravity
\[ \alpha \] pitching (torsion) displacement, posi-
tive nose up
\[ \alpha_a \] angle of attack, deg
\[ \eta \] normal direction of transformed coordi-
nate system
\[ \mu \] mass ratio
\[ \zeta_h \] critical damping coefficient in
plunging
\[ \zeta_a \] critical damping coefficient in
pitching
\[ \xi \] chordwise direction of transformed
coordinate system
\[ \rho \] air density
\[ \omega_n \] uncoupled plunging frequency, \( \sqrt{K_h/m} \)
\[ \omega_a \] uncoupled pitching frequency, \( \sqrt{K_a/I_a} \)

Superscripts
\[ (\cdot) \cdot \] \( d(\cdot)/dt \)
\[ (\cdot)'' \] \( d(\cdot)/d\xi \)
\[ n-1, n, n+1 \] time levels

Introduction

The requirements of high aerodynamic effi-
ciency and low noise in the operating range of
transonic flow has resulted in thin (2 to 4 percent
thick), highly swept and twisted blades for prop-
fans. These blades operate at moderate to large
mean angles of attack. In wind tunnel tests of a
ten-bladed, highly swept SR5 propfan model, con-
ducted at NASA Lewis Research Center, the blades
have fluttered at transonic tip Mach numbers at sea
level conditions. The aeroelastic characteristics
of this model were investigated in Refs. 1 to 3 by
using a two-dimensional linear subsonic unsteady
cascade aeroelastic theory with a correction for
blade sweep. The correlation between theory and
experiment varied from poor to good, depending on
the test parameters. To further understand the
physics of the flutter phenomena and to validate
the recently developed linear cascade aeroelastic
models for three-dimensional subsonic flow condi-

\[ a \]
Flutter calculations using Euler equations were presented in Ref. 23. The study showed significant differences in the flutter boundaries, compared with those obtained using TSD theories, when the steady shocks are strong and at or near the trailing edge. Reference 16 presented steady and unsteady pressure calculations using Euler equations, and concluded that "the interaction of shock with the boundary layer is strong enough to alter the airfoil pressure distribution substantially." The interaction of a shock and boundary layer may lead to boundary layer separation resulting in a flow condition which is totally beyond the capability of Euler equations. Also, the Euler equations do not show the effect of viscosity on the flutter boundary. This means that for most accurate prediction of the aerodynamic loads and the flutter boundary, solution of the Navier-Stokes equations is needed. It would also be noted here that the calculations done so far in the published literature have been with 6 to 12 percent thick airfoils. With the advent of propfans, consisting of 2 to 4 percent thick airfoils, an investigation is required for thinner airfoils.

In the present paper, the Euler/Navier-Stokes flow solver, which is described in Ref. 26, and coupled to a binary structural dynamic model in Ref. 24, will be applied first to study the effects of rotational flow, initial conditions, mean angle of attack, viscosity, blade shape, and thickness on the transonic flutter characteristics of a swept NACA 64A010 airfoil with the structural parameter, given in Ref. 20. Next, the coupled binary structural dynamic model flow solver will be applied to a thin, swept SR5 propfan blade section to study its transonic flutter characteristics. It is to be mentioned here that the artificial dissipation model used in Ref. 24 was based on the local pressure gradient scaled by a constant factor. Recently, efforts are being made to incorporate a second dissipation model in this code. This model is based on the local pressure gradient scaled by a spectral radius. Preliminary results obtained using this model will also be presented for comparison.

**Aeroelastic Model**

A typical section model of a blade, as shown in Fig. 1(a), is used for the flutter study. The typical section has two degrees of freedom, plunging \( h \) and pitching \( \alpha \), positive as shown. The governing equations for this model are

\[
\ddot{h} + \frac{1}{2} \alpha + C_h h + K_h \dot{h} = Q_h
\]

\[
\dddot{\alpha} + \frac{1}{2} \alpha + C_\alpha \dot{\alpha} + K_\alpha \alpha = Q_\alpha
\]

where \( m \) is the mass, \( S \) is static unbalance, \( C_h, C_\alpha \) are structural damping parameters, \( K_h \) and \( K_\alpha \) are the plunging and pitching spring constants, \( h \) and \( \alpha \) are the generalized forces, and \( \dot{\alpha} \) is \( \frac{d\alpha}{dt} \). Defining \( \ddot{h} = h/2b, x_\alpha = S/mb, r_\alpha = I_\alpha /

\[
h, 0 = \frac{K_h}{m}, \omega_h = K_\alpha / \alpha, C_h = 2C_h/m, \text{ and } C_\alpha = 2C_\alpha/m/\alpha.\]

\[
C_h = 2C_h \omega_h, \text{ and } C_\alpha = 2C_\alpha \omega_\alpha, \text{ where } b \text{ is the semi-chord, } I_\alpha \text{ is the mass moment of inertia about the elastic axis, } \omega_h \text{ and } \omega_\alpha \text{ are the uncoupled plunging and pitching frequencies, } C_h \text{ and } \omega_\alpha \text{ are the critical damping factors in bending and torsion, Eq. (1), can be written as}
\]

\[
\ddot{h} + \left[ \frac{\alpha}{2} \right] \dot{h}_{\alpha} + \left[ 2C_h \omega_h \right] h = 0
\]

\[
\frac{\omega_h}{2} h = 0
\]

The generalized forces, \( Q_h \) and \( Q_\alpha \), are related to the lift and moment coefficients, obtained from the aerodynamic code as follows:

\[
Q_h = -\frac{1}{2 \rho V^2(2b)} c_1
\]

\[
Q_\alpha = \frac{1}{2 \rho V^2(2b)} c_m
\]

where \( c_1 \) is the lift coefficient, and \( c_m \) is the moment coefficient about the axis of rotation (elastic axis). \( V \) is the resultant velocity and \( \rho \) is the air density. Defining, \( \dot{\alpha} = \alpha(2b) \), and \( \dddot{\alpha} = \alpha(2b) \), where \( \dot{\alpha} \) is the speed of sound and substituting Eq. (3) into Eq. (2), the final aeroelastic equations can be written as

\[
\dddot{h} + \left[ \frac{\alpha}{2} \right] \dot{h}_{\alpha} + \left[ 2C_h \omega_h \right] h = 0
\]

\[
\dddot{\alpha} + \left[ \frac{\alpha}{2} \right] \dot{\alpha}_{\alpha} + \left[ 2C_\alpha \omega_\alpha \right] \alpha = 0
\]

\[
\left[ \frac{\alpha}{2} \right] \dddot{h} + \left[ \frac{\alpha}{2} \right] \dot{h}_{\alpha} + \left[ \frac{\alpha}{2} \right] \dot{\alpha}_{\alpha} + \left[ \frac{\alpha}{2} \right] \dot{\alpha}_{\alpha} + \left[ \frac{\alpha}{2} \right] \dot{\alpha}_{\alpha} = 0
\]

where \( \dddot{h} = \left( h^{(n+1)} - h^{(n)} \right) / \Delta t \) and \( \dot{\alpha}_{\alpha} = \left( \alpha^{(n+1)} - \alpha^{(n)} \right) / \Delta t \). Substitution of these finite difference derivatives for \( h, \dot{h}, \alpha, \dot{\alpha} \) into Eq. (4) leads to a system of simultaneous equations for \( h \) and \( \alpha \) at each time step.

\[
\left( h^{(n+1)} - h^{(n)} \right) / \Delta t = \left( h^{(n+1)} - h^{(n)} + h^{(n-1)} \right) / (2 \Delta t)
\]

\[
\left( \alpha^{(n+1)} - \alpha^{(n)} \right) / \Delta t = \left( \alpha^{(n+1)} - \alpha^{(n)} + \alpha^{(n-1)} \right) / (2 \Delta t)
\]

The lift and moment coefficients \( (c_1, c_m) \) in Eq. (4), are calculated by integrating the pressure distribution over the airfoil surface at each time step. The pressure is obtained by solving the unsteady, two-dimensional, Reynolds-averaged, compressible Euler or Navier-Stokes equations on a body-fitted coordinate system in strong conservation form using an alternate direction implicit (ADI) procedure. The formulation has been high. Even though the flutter boundaries have been predicted using these methods, the question has always been asked, how strong are the effects of rotational flow, shock location, and shock strength on the prediction of flutter boundaries? These effects can only be studied by using either the Euler equations or the Navier-Stokes equations.
described in Ref. 26, and only a brief outline is given here. All the calculations are performed in a body-fitted coordinate system (ξ, η, τ) which is mapped to the Cartesian coordinates (x, y, t). Fig. 1(b), according to the following one-to-one relationship:

\[ ξ = ξ(x, y, t) \]
\[ η = η(x, y, t) \]
\[ τ = t \]  

(6)

The Jacobian of the transformation J is given by

\[ J = J = \frac{1}{(x_ξ y_ξ - x_η y_η)} \]  

(7)

and the metrics of the transformation are given by the relationship:

\[ E_x = J_y y_ξ - J_x x_η ; \quad E_y = J_x x_ξ - J_y y_η ; \quad E_τ = J_x x_ξ - J_y y_η \]  

(8)

Once a grid has been constructed the metrics of the transformation can be evaluated numerically. Standard central differences were used to compute the quantities such as \( S_x, S_y, \) etc. At the boundaries, three-point one-sided differences were used to compute the metrics.

In the (ξ, η, τ) coordinate system, the two-dimensional unsteady Navier-Stokes equations may be written as

\[ \frac{∂q}{∂t} + \nabla \cdot F = 0 \]  

(9)

where

\[ q = (p, ρu, ρv, e) \]

and \( ρ \) is the fluid density; \( u \) and \( v \) are the Cartesian components of fluid velocity; \( e \) is the total energy of the fluid per unit volume. The quantities \( F, G, R, \) and \( S \) are given by

\[ F = \frac{(E_x F + E_y G + E_τ q)}{J} \]
\[ G = \frac{(E_x G + E_y S + E_τ q)}{J} \]
\[ R = \frac{(E_x R + E_y S)}{J} \]
\[ S = \frac{(E_x S)}{J} \]  

(10)

The terms \( F \) and \( R \) are the flux and viscous stress terms along the \( x \)-direction and \( G \) and \( S \) are likewise the flux and viscous terms along the \( y \)-direction and are given as follows:

\[ F = (p u, p u^2 + p, p u v, u (e + p)) \]
\[ G = (p v, p u v, p v^2 + p, v (e + p)) \]
\[ R = (0, τ_{xx}, τ_{xy}, R_q) \]
\[ S = (0, τ_{xy}, τ_{yy}, S_q) \]  

(11)

The quantities \( R_q \) and \( S_q \) represent the dissipation of energy due to work done by the fluid in the \( x \)- and \( y \)-directions respectively. The viscous stresses \( τ_{xx}, τ_{xy}, \) and \( τ_{yy} \) are related to the velocity gradients through Stokes' hypothesis.

Since the governing equations, Eq. (9), are coupled and highly nonlinear, a stable and efficient solution procedure is required. In the present work, the Beam-Warming algorithm,28 as implemented by Steger,29 was followed with modifications noted by Sankar and Tang.26 The viscous terms are evaluated explicitly to reduce the number of operations needed to solve the governing equations. A brief description of the solution scheme is given here.

The governing equations are written at a computational node \((i, j)\) in the following finite difference form:

\[ \frac{∂q_{i,j}^{n+1}}{∂t} + \frac{∂F_i}{∂x} + \frac{∂G_j}{∂y} = 0 \]  

(12)

where

\[ ∂F_i/∂x \] and \( ∂G_j/∂y \) are standard central difference operators. For example, the term \( ∂F_i/∂x \) is the standard two point central difference formula given by \((F_{i+1,j} - F_{i-1,j})/2\). \( D_{ij} \) is an artificial dissipation term and is discussed in the next section.

The highly nonlinear terms \( F \) and \( G \) at the time level \((n+1)\) were expanded by a Taylor series about a previous time level \( n \) as

\[ F_{i,j}^{n+1} = F_{i,j}^n + \left( \frac{∂F_i}{∂q} \right)_n q_{i,j}^{n+1} \]
\[ G_{i,j}^{n+1} = G_{i,j}^n + \left( \frac{∂G_j}{∂q} \right)_n q_{i,j}^{n+1} \]  

(13)

where \( ∂F/∂q \) and \( ∂G/∂q \) are 4 by 4 matrices which are the Jacobians of the flux terms \( F \) and \( G \) with respect to \( q \).

In order to allow large values of the explicit dissipation coefficient \( c_E \) to be used without instability, and to allow the viscous terms to be treated explicitly, the following implicit dissipation terms were added to the left side of the difference Eq. (13).

\[ -c_F \frac{1}{J} (E_x G + E_y R) ∆q_{i,j}^{n+1} \]  

(14)

where \( J \) is the Jacobian of the transformation. The coefficient \( c_F \) was taken to be two to three times the explicit dissipation coefficient \( c_E \). A range of \( c_E \) values between 2 and 5 were used in the calculations.

The dissipative term, \( D_{ij} \), is added to remove high-frequency errors in the solution at every time step, and to avoid odd-even decoupling of the numerical solutions. Two models of introducing artificial dissipation are available in the present code. In model II,26 the dissipation term is written, in conservative form, as a combination of second and fourth order dissipative terms. This model
is based on the local pressure gradient scaled by a constant factor. A sensor based on the second derivative of pressure, turns the second order dissipation term in the vicinity of shocks and suppresses the fourth order dissipation terms. This sensing is followed only in the streamwise direction. A more detailed description of this model, and bench mark calculations are presented in Ref. 26. This model, as pointed out in Ref. 30, adds positive dissipation in certain regions and negative dissipation in other regions depending on the flow field gradients in that region, which may degrade the solution accuracy in some flow problems.

In model II, the local pressure gradient is scaled with the spectral radius of the flux vector. The fourth order difference term is modified in a way that it produces positive dissipative terms. This model is similar to the one presented in Refs. 31 and 32 and is being currently implemented in the present code by Wu. Some preliminary test results will be given in the present paper.

Equation (13) may be written after the addition of the artificial implicit dissipation terms given by Eq. (15), in the following operator form

\[ \Delta q^{n+1} = R^n \]  

where

\[ R^n = -\Delta t(\xi\xi' + \eta\eta')^n + \Delta t(\xi\eta + \eta\xi')^n - \Delta t e_2^2 \]  

The left hand side operator of Eq. (16) was approximately factored into two small operators, leading to the following final form:

\[ \left[ 1 + \Delta t \frac{\partial P}{\partial q} - e_1 \Delta t J^{-1} \xi \xi' \right] \]  

\[ \times \left[ 1 + \Delta t \eta \frac{\partial G}{\partial \eta} - e_1 \Delta t J^{-1} \eta \eta' \right] \Delta q^{n+1} = R^n \]  

Equation 18 may be solved through the inversion of two block tri-diagonal matrix equations, one corresponding to the \( \xi \)-direction, and the other corresponding to the \( \eta \)-direction. In order to keep the flow solver simple, the boundary conditions on all the boundaries were explicitly updated after the interior points had been updated using Eq. (18).

The Baldwin-Lomax eddy viscosity model is used to model the turbulent momentum and energy transfer. This is a two layer algebraic turbulence model with two algebraic equations in each layer.

Algebraic grid generation routines were included by Sankar and Tang to make the code more portable. Both O- and C-grids can be generated. However, any grid can be overwritten for the algebraic grid, for example, by a numerical grid.

The boundary conditions were treated explicitly as follows: at the solid surface, the no slip boundary conditions were enforced by setting the velocity of the fluid to that of the solid surface at every time step. At the solid surface, the normal derivative of the temperature and pressure were also set to zero. The use of the C-grid system introduces a cut between the airfoil trailing edge and the downstream boundary. Along this cut, the flow properties were averaged from above and below. The body-fitted grids employed in this work were such that the far field boundaries were at least six chord lengths away. The flow properties at the far field boundaries were assumed to be undisturbed.

The Navier-Stokes solver described here may also be used to model inviscid rotational flows. This requires that the viscous terms in the Navier-Stokes equations be suppressed, and that the zero tangential velocity boundary condition at the solid surface be replaced by a nonzero value, obtained by extrapolating the tangential component of the fluid velocity from the interior.

The code is expected to predict the flow field for arbitrary airfoils for moderate angles of attack, a Mach number range from 0.1 to 1.2, and a Reynolds number range from 100 to 10^7. This code has been used to predict the dynamic stall characteristics for the NACA 0012 airfoil, transonic flow solutions, and stall flutter characteristics for the NACA 0012 airfoil.

Results and Discussion

This section presents the numerical results obtained with the aeroelastic model described earlier. First, the code is validated with published results. Next, the effects of rotational flow, initial conditions, mean angle of attack, and shape and thickness on transonic flutter are investigated. Then transonic flutter calculations are done for the SRS propfan blade typical section model.

Code Validation

References 24, 26, and 36 presented selected validation cases with the present Euler/Navier-Stokes flow solver. These cases represented different airfoils at different flow conditions. In those references, the dissipation model I, which was described earlier was used. As a part of the validation study of the dissipation model II, described earlier, the pressure distribution for RAE2822 airfoil is recalculated. The results are compared with those obtained in Ref. 36 with dissipation model I and the experimental data presented therein. Figure 2 shows the steady pressure distribution (pressure coefficient, \( C_p \), versus non-dimensional distance along the chord, \( \chi = 0.5(x/b + 1) \), see the coordinate system in Fig. 1(a)); for the RAE2822 airfoil at Mach number, \( M = 0.73 \), angle of attack, \( \alpha = 2.79^\circ \); and Reynolds's number, \( Re = 6.5 \) million. The numerical results with both dissipation models and the experimental data show very good comparison. These comparisons help to give confidence in the code and in the attempt at flutter calculations with the dissipation model II.

In order to establish the capability of the code for predicting the transonic flutter dip, the flutter boundary of the NACA 64A010 airfoil is obtained. The calculations are carried out using
the Euler version of the code with both dissipation models. A 157 by 40 algebraic C-grid shown in Fig. 3 is used in the calculations. The structural parameters of the blade are shown in Fig. 20 and are given as case A in Table 1. In Ref. 20, the wing geometric sweep is modeled by positioning the elastic axis ahead of the leading edge (a_h = -2.0, see Fig. 1(a)).

To obtain the flutter boundary, a steady-state flow field solution is first obtained for 0° mean angle of attack. The solution of the aeroelastic equations (Eq. (4)) is then started at an assumed nondimensional flutter speed, V* (V/V_{	ext{inf}}), by prescribing the initial conditions, h(0), a(0), h(0), and a(0). If the amplitude of oscillation of the resulting transient solution increases (decreases), the value of V* is reduced (increased) and the transient solution is again calculated. The critical flutter speed (V_{*\text{c}}) is obtained by linearly interpolating the logarithmic decrement of the amplitude of oscillations for two different but close values of V* (one giving oscillations with increasing amplitude and the other giving oscillations with decreasing amplitude). The flutter boundary is presented as a variation of the flutter speed index (V_{*\text{c}}/V) versus Mach number (M).

The results obtained for the NACA 64A010 airfoil are compared with published results10,11,21,23 in Fig. 4. Refs. 10, 11, and 21 used transonic small disturbance (TSD) theory and Ref. 23 used Euler equations. In this comparison, differences due to grids, numerical schemes, artificial viscosity, etc., employed in the published results are not considered but only the flutter boundaries are compared. In the present calculations, the initial conditions, a(0) = 0.1°, with zero h(0), h(0), and a(0) are used. A comparison shows that all of the codes10,11,21,22 qualitatively predict the transonic dip. However, the transonic recovery portion of the flutter boundary predicted by the Euler codes differs from those predicted by codes based on TSD theory. The recovery predicted by Euler codes occurs at a higher Mach number compared to the one predicted by TSD codes. Also, the Euler code of Ref. 23 predicts the recovery to be at a higher Mach number compared to the present Euler code. Beyond M = 0.88, the shock strength is sufficiently high enough to induce separation. The Euler and TSD codes fail to model separated flow behind shocks and predict qualitatively the same flutter boundary beyond M = 0.88.

Also, there are significant qualitative differences in the recovery region of the flutter boundary obtained by the two dissipation models available in the present Euler code. The dissipation model I predicts a flutter boundary which folds over to provide upper boundaries. For a Mach number of 0.9, three boundaries are observed. This compares qualitatively with the boundaries obtained in Refs 10, 11, and 23. Whereas no such folding over of the boundary is observed when dissipation model II is used. Between Mach numbers of 0.8 and 0.9 the flutter boundary jumps considerably, and is almost the same as the third boundary obtained with dissipation model I. This has also been observed by Isogai.21 Again, this difference can be explained by observing the pressure distribution obtained with the two dissipation models. The pressure distributions obtained with M = 0.9 with the two dissipation models are shown in Fig. 5. Both models show considerable differences with regard to the shock location and shock strength. The dissipation model II predicts the shock to be closer to the trailing edge than that predicted by dissipation model I. This may be the reason for the difference observed in the recovery region with these models.

Figure 6 shows a typical transient response solution. Figure 6(a) shows the transient response for a point below the flutter speed and Fig. 6(b) shows the response for a point above the flutter speed. In both the responses, the amplitude of h is larger than that of a, indicating possible bending mode instability.

The above comparisons for the NACA 64A010 help to give confidence in the code for investigation of the flutter dip phenomena for other airfoils. It also shows the importance of the accurate prediction of the shock strength and position.

The code is now applied to study the effects of the following: (1) initial conditions, (2) mean angle of attack, (3) viscosity, and (4) thickness and shape. Dissipation model I is used in all the calculations.

Effect of initial conditions. To study the effect of initial conditions, the initial condition on a(0) is increased from 0.1° to 4°. The flutter boundary for 4°, together with that for 0.1°, is shown in Fig. 7. For a Mach number of 0.8 the flutter speed for 4° decreases by approximately 30 percent. This has also been observed in Ref. 23. However, for Mach numbers of 0.85 and 0.88 there does not seem to be any significant change in flutter speeds (less than 10 percent). This again seems to be due to the position of the shock. Since the shock is near the trailing edge of the airfoil, the effect of rotation does not...
decrease in thickness which delays the formation of the shock to higher Mach numbers. Because of smaller thickness, the shock strength is also small relative to that of NACA 16-010. This results in a smaller jump in the flutter boundary in the recovery region. For the same Mach number, the NACA 16-004 airfoil has a shock appearing behind that of the NACA 16-010 airfoil which results in a smaller region of rotational flow for the NACA 16-004 airfoil. The combined effect of the shock location and shock strength, and rotational flow is to lower the boundary for the NACA 16-010 as compared to that of the NACA 16-004 for the same Mach number. The flutter boundaries calculated up to M = 0.9 for the NACA 64A010 and the NACA 16-010 airfoils did not show any difference, indicating thickness has more effect on the flutter boundary than the shape.

The flutter boundary obtained for the NACA 16-(1.3)(04) is also shown in Fig. 8. The pressure distribution curves, not shown here, for this airfoil showed a shock at a Mach number equal to 0.85, which resulted in a drop in the flutter speed index compared to that for the NACA 16-004 airfoil. The shock is at the trailing edge beyond M = 0.88, and the flutter speed index starts increasing again. The similarity of the flutter boundary curve with that of the NACA 16-004 airfoil is consistent with the observation that the camber effect is equivalent to the effect of initial angle of attack for the NACA 16-(1.3)(04) airfoil.

In Fig. 8, the flutter boundary obtained for the NACA 16-(1.3)(2.6) airfoil (the propfan airfoil) is also shown. A dip in the flutter boundary can be seen near M = 0.925. There after the flutter speed index starts increasing. It is interesting to note that the flutter boundary between Mach numbers 0.85 and 0.95 is almost similar to that obtained for the NACA 16-004 airfoil. This is due to the fact that the reduction in the flutter speed index due to camber, like that for the NACA 16-(1.3)(04) airfoil, is compensated by a decrease in thickness. Between Mach numbers 0.75 and 0.85, the flutter boundary for the NACA 16-004 and the NACA 16-(1.3)(2.6) airfoils show the effect due to thickness, as was observed for the NACA 16-010 and the NACA 16-004 airfoils. A typical response for the NACA 16-004 is shown in Fig. 9. Figure 9(a) shows the transient response for a point below the flutter speed, and Fig. 9(b) shows the response for a point above the flutter speed. In both responses, the amplitude of \( h \) is larger than that of \( \alpha \), indicating possible bending mode instability.

The comparison of the results from the four airfoils shows that (1) for symmetric airfoils, the transonic dip moves towards higher Mach numbers as thickness to chord ratio increases. For example, the thin cambered airfoil studied here, the camber effects are nullified by the effects due to reduction in thickness, and the flutter boundary is similar to a thick symmetric airfoil of the same series, and (3) the flutter characteristics strongly depend on the shock location and shock strength.

After establishing confidence in the code, it was applied to investigate the flutter characteristics of a simulated SR5 propfan blade.
Flutter Characteristics of a Simulated SR5 Propfan Blade

A typical section model is constructed which simulates the structural properties, mode shapes, and coupled frequencies of the SR5 propfan. Linear subsonic isolated airfoil theory is used in simulating the typical section model. The rotation axis (elastic axis) and the uncoupled bending and torsional frequencies are chosen such that the mode shapes and coupled frequencies are close to those calculated for a SR5 propfan blade using NASTRAN analysis. The selected properties are shown in Table I, case B. It should be noted here that the frequency ratio is almost equal to one as in case A, studied earlier, and the mass ratio is approximately twice that of case A. For the structural properties selected, the calculated flutter Mach number and flutter reduced frequency are 0.875 and 0.17. The corresponding experimental values are 0.88 and 0.17 respectively.

The flutter boundary obtained using the present Euler code with dissipation model is shown in Fig. 10. The flutter boundary obtained with linear isolated blade theory is also shown in Fig. 10 along with the velocity index curve \((V/\sqrt{BD}_{0.2} \mu)\) versus Mach number. The present Euler code gives a flutter Mach number of 0.845. This value is about 4.5 percent less than that obtained from SR5 propfan experiment. The prediction of lower flutter Mach number by the present Euler code may be due to the following reasons: (1) the limitation of the typical structural dynamic model employed to simulate the SR5 propfan and (2) no structural damping is included in the present analysis.

Figure 10 shows that the flutter speed index drops as the Mach number increases to \(M = 0.9\), thereafter, it starts recovering, showing a very low transonic flutter dip for SR5 propfan blade. The Euler code predicts almost the same flutter boundary as the linear isolated blade theory. However, much more pronounced transonic dip is expected when the following aspects are considered. The above calculations are made at zero mean angle of attack. In a real situation, the problem is at a mean angle of attack which may cause transonic flutter dip. The mass ratio used is 115, which is higher than for the present composite propfans having the mass ratio of the order of 33 to 60. This low mass ratio may cause transonic flutter dip which needs further investigation. The cascading effect which is found to have destabilizing effect on flutter in subsonic flow, may also affect the transonic dip.

The following points support the above points when the flutter boundary obtained for the NACA 16-(1.3)(2.8) airfoil in Figs. 8 and 10 are compared. In Fig. 8, a relatively large dip is seen for the same airfoil section. Here the mass ratio is 60, which is about 48 percent less than that used for SR5 propfan blade typical section in Fig. 10. Also, the elastic axis position is one semichord length in front of the leading edge compared to that used for SR5 propfan blade typical section. Since here only an attempt is made to simulate a SR5 propfan, these structural properties need to be improved for correct prediction of flutter boundary and correlation with experiment in transonic flow for thin highly swept propfans.

Conclusions

An Euler/Navier-Stokes flow solver is used to obtain the flutter boundaries for a typical section structural model with several airfoil sections. The application of the code for predicting transonic flutter characteristics of thick and thin airfoils is demonstrated. Based on this study, the following conclusions are drawn.

1. The rotational flow effects behind the shock have a strong effect on the transonic flutter speed depending on the chordwise location of the shock. The neglect of rotational flow effects results in predicting a higher flutter speed.

2. The two dissipation models employed, one based on the local pressure gradient scaled by a constant factor and the other based on the local pressure gradient scaled by spectral radius, predicted the same flutter boundary up to the transonic dip for a thick airfoil. However, in the recovery region, the boundaries predicted by these models are different.

3. The effects of initial conditions, mean angle of attack, and viscosity for the Reynolds number studied on the minima of the transonic dip seems negligible. However, they have a significant effect away from the dip.

4. The blade thickness and shape dictate the location and strength of the shock, there by affecting the flutter boundary. The transonic dip shifts to higher Mach numbers for symmetric airfoils with decreasing airfoil thickness to chord ratio. The flutter boundary, for relatively thin cambered airfoils studied here, showed a relatively low transonic dip compared to a symmetric thick airfoil.

5. The predicted flutter Mach number for a simulated SR5 propfan blade is about 4.5 percent less than that obtained by experiment. This difference is attributed to the simplified aeroelastic model used in the present analysis. The flutter boundary showed a very low transonic dip. Further studies are needed to investigate the effects of mean angle of attack, mass ratio, and elastic axis position on the transonic flutter of highly swept composite propfans.

Acknowledgments

During the course of this investigation, the first author has been supported by NASA Grant NAG-3139 and the second author by NAG 3-730. The authors would like to thank Prof. N.L. Sankar, and Mr. J.C. Wu of Georgia Tech and Mr. Dennis Huff of NASA Lewis Research Center for many helpful discussions.

References


Table 1. - STRUCTURAL PARAMETERS USED IN THE STUDY

[Distsances are nondimensionalized with respect to semi-chord (b), see Fig. 1(a) for definitions.]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Case A</th>
<th>Case B</th>
</tr>
</thead>
<tbody>
<tr>
<td>a/h</td>
<td>-2</td>
<td>-1</td>
</tr>
<tr>
<td>a/b/a_0</td>
<td>1</td>
<td>0.928</td>
</tr>
<tr>
<td>a_0</td>
<td>1.8</td>
<td>0.9644</td>
</tr>
<tr>
<td>Tcg</td>
<td>0.4898</td>
<td>0.4769</td>
</tr>
<tr>
<td>μ</td>
<td>60</td>
<td>115</td>
</tr>
<tr>
<td>C_h/C_α</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
(A) TWO-DEGREES OF FREEDOM TYPICAL SECTION MODEL.

(B) PHYSICAL AND COMPUTATIONAL DOMAINS.

FIGURE 1. - COORDINATE SYSTEMS FOR STRUCTURAL AND AERODYNAMIC MODELS.
PRESENT (DISSIPATION MODEL II)
PRESENT (DISSIPATION MODEL I, REF. 36)
\[ M = 0.730, \theta = 2.79^0 \]

EXPERIMENT (REF. 36)
\[ M = 0.725, \theta = 2.92^0 \]

Figure 2. Pressure distributions for RAE 2822 airfoil, \( Re = 6.50 \) million. (157\times58 numerical C-grid.)
FIGURE 3. - PORTION OF AN ALGEBRAIC C-GRID SYSTEM AROUND NACA 64A010 AIRFOIL.

FIGURE 4. - FLUTTER BOUNDARIES FOR THE NACA 64A010 AIRFOIL. (CASE A, TABLE 1.)
(A) $M = 0.90, \alpha_a = 0$, EULER (DISSIPATION MODEL I).

(B) $M = 0.90, \alpha_a = 0$, EULER (DISSIPATION MODEL II).

FIGURE 5. - STATIC PRESSURE DISTRIBUTION, NACA 64A010 AIRFOIL.
FIGURE 6. - PLUNGING AND PITCHING AMPLITUDES VERSUS TIME. NACA 64A010 AIRFOIL, $M = 0.88$, $a_h = -2.0$, $\omega_h/\omega_g = 1.0$, $x_a = 1.8$, $r_a = 1.865$, EULER, ALGEBRAIC C-GRID (157x40).
MEAN ANGLE OF ATTACK
0° INITIAL PERTURBATION
2° MEAN ANGLE OF ATTACK
Re = 12x10^6

FIGURE 7. - FLUTTER BOUNDARIES FOR THE NACA 64A010 AIRFOIL. (CASE A, TABLE 1.)
FIGURE 8. - FLUTTER BOUNDARIES FOR FOUR AIRFOILS.
(CASE A, TABLE 1, DISSIPATION MODEL I.)
FIGURE 9. - PLUNGING AND PITCHING AMPLITUDES VERSUS TIME, NACA 16-004 AIRFOIL. $M = 0.92$, $\zeta_h = -2.0$, $\omega_h/\omega_a = 1.0$, $x_a = 1.8$, $f_a = 1.865$, EULER, ALGEBRAIC C-GRID (157x40).
FIGURE 10. - FLUTTER BOUNDARY FOR SR5 PROPFAN SIMULATED TYPICAL SECTION MODEL. (CASE B, TABLE 1, DISSIPATION MODEL I.)
The transonic flutter dip phenomena on thin airfoils, which are employed for propfan blades, is investigated using an integrated Euler/Navier-Stokes code and a two degrees of freedom typical section structural model. As a part of the code validation, the flutter characteristics of the NACA 64A010 airfoil are also investigated. In addition, the effects of artificial dissipation models, rotational flow, initial conditions, mean angle of attack, viscosity, airfoil thickness and shape on flutter are investigated. The present results obtained with a Euler code for the NACA 64A010 airfoil are in reasonable agreement with published results obtained by using transonic small disturbance and Euler codes. The two artificial dissipation models, one based on the local pressure gradient scaled by a common factor and the other based on the local pressure gradient scaled by a spectral radius, predicted the same flutter speeds except in the recovery region for the case studied. The effects of rotational flow, initial conditions, mean angle of attack, and viscosity for the Reynolds number studied seem to be negligible or small on the minima of the flutter dip. However, they have significant effect on the flutter boundary away from the dip. The flutter dip shifts towards higher Mach number as the thickness decreases for symmetrical airfoils, other parameters being the same. This is in direct relation to the location and strength of the shock. The flutter boundary for a thin cambered airfoil (propfan airfoil) showed that the effect of camber is nullified by the effect due to reduction in thickness and showed a relatively low transonic dip. The flutter boundary of a simulated SR5 propfan typical section model showed a very low transonic flutter dip. However, further studies with varying mean angles of attack and mass ratio are required to better understand the transonic flutter dip phenomena of highly swept propfans.