Numerical modeling of flutter in a transonic fan
O. MILANDRE
Royal Institute of Technology, 10044 Stockholm, Sweden

Flutter is a self-feeding and potentially destructive vibration that can lead to devastating effects such as broken blades. Using accurate numerical models to predict flutter in the conception of an engine is essential to avoid huge waste of money. The software 2, using the elsA CFD package developed by the French Aerospace Lab, ONERA, is used to perform unsteady calculations and predict flutter margin. The current methodology does not systematically manage to reproduce the expected flutter pocket in transonic areas. The aim of the study is to investigate the impact of some parameters, such as boundary conditions and the position of the inlet and outlet plans of the mesh, on unsteady results. Results show that the current methodology, using conditions of injection at the inlet and an association of radial equilibrium and valve law boundary at the outlet, is inaccurate due to problem of reflecting waves that disturb the results. Using conditions of non-reflection at the inlet and the outlet seems more appropriate. Nevertheless, the new flutter margin using conditions of non-reflection does not permit to reproduce the flutter pocket either. New lines of investigation such as the impact of the density of the mesh or the time scheme may improve the numerical results.

Introduction

The flutter phenomenon, caused by the coupling between the vibration of the structure on a nodal mode and the unsteady aerodynamic forces created by the vibration itself, may appear on the fan of an aeroengine. Given that flutter is a potentially destructive vibration due to its divergent nature, it has to be taken into consideration during the conception of a fan or plane in order to avoid any serious damage or structural failure, as it occurred on Braniff Flight 542.

The ability to accurately predict the occurrence of flutter using computational tools is a major stake in the conception of an engine nowadays, because the tendency of new aeroengines is a decrease of the weight and an increase of the load, hence more significant blade interactions and reduced flutter margins. Any error may induce huge financial losses by increasing design costs. The origin of flutter depends on the operating range of the engine.

Transonic flutter that appears for high intake mass flow in a fan is studied since this phenomenon is more difficult to simulate than, for example, subsonic flutter. The cause of the phenomenon is still unclear and it involves non-linear aerodynamic phenomena such as shock waves and boundary layer separations.

One engine is considered. The transonic flutter of a blade has already been studied with the software 1 using the elsA CFD package developed by the French Aerospace Lab, ONERA. The computational results did not succeed in reproducing the sensitivity of flutter in transonic areas, as observed on test beds. A new software, referred to as software 2, has been since developed using the same CFD solver.

The aim of the study is first to assess software 2 on steady and unsteady computational results using the same methodology. Then alternative numerical settings are tested in order to improve the results and better reproduce the flutter margin observed on test beds.

The flutter phenomenon

The flutter phenomenon is a manifestation of dynamic aeroelasticity which studies the interaction between elastic forces, aerodynamic forces and inertial forces on any given elastic structure.

a. Self-sustained asynchronous phenomenon

Flutter is a self-feeding phenomenon caused by the coupling between structure and its surrounding flow field, which frequency does not depend on the rotational speed of the engine. Its response involves one or many natural modes of the blade. Just before the appearance of flutter, the total damping of the aeroelastic system is equal to zero, what
means that some oscillations caused by any perturbation will remain. The self-sustained event is due to an exchange of energy between the structure and the flow.

Given that transonic flutter is studied on a fan, an empirical assumption can be made. Flutter on a transonic fan only occurs on one natural mode, and therefore may be considered as an harmonic phenomenon.

![Figure 1: Compressor Map [1]](image)

The transonic flutter may occur on an engine for high pressure ratios and mass flows (index 2 on Figure 1). High engine speed causes the apparition of a shock on the leading edge of the blade. The structure may become unstable because of the interaction of the shock and the boundary layer, or the oscillation of the shock itself.

a. Aerodynamic damping

The aim of the study is to accurately model transonic flutter on a fan. The most significant parameter to determine the stability of a blade is to calculate total damping, which is equal to the sum of the aerodynamic damping and the structural damping.

\[
\delta_{\text{total}} = \delta_{\text{aero}} + \delta_{\text{mechanical}}
\]

(1)

Where \( \delta_{\text{aero}} \) and \( \delta_{\text{mechanical}} \) are the aerodynamic and the structural damping, respectively.

\( \delta_{\text{mechanical}} \) stands for the energy dissipated by the structure. It is always positive, and thus stabilizes the structure by damping the vibrations. The damping force is partly due to rubbing surfaces, friction. \( \delta_{\text{mechanical}} \) is alleged to be constant in our study.

\( \delta_{\text{aero}} \) [1] [2] is defined as follows:

\[
\delta_{\text{aero}} = \frac{W}{\mu \omega^2} = -\int_0^T \int p(t,M)\bar{n}.V(t,M)\bar{e}dt\,dS
\]

(2)

Where \( W \), \( p \), \( V \), \( \mu \), \( \omega \), \( \bar{e} \) are the aerodynamic work, the unsteady pressure, the velocity of the motion, the modal mass, the modal frequency and the vector collinear to the motion, respectively.

As the motion is harmonic, \( W \) can be rewritten as:

\[
W = -\int \pi. A_m. A_p. \sin(\varphi) \bar{n}\bar{e}dS
\]

(3)

Where \( A_m \), \( A_p \), \( \sin(\varphi) \), \( \bar{n}\bar{e} \) are the amplitude of the motion, the amplitude of the first harmonic of the unsteady pressure, the phase shift between the unsteady pressure and the motion and the vector perpendicular to the surface of the blade pointed outwards, respectively.
\( \delta_{\text{aero}} \) is proportional to the aerodynamic work (2). During each cycle of vibration, some aerodynamic work is exchanged with the blade motion. If \( \delta_{\text{mechanical}} \) is not considered, a negative aerodynamic work means that the blade vibration will increase: the blade absorbs energy from the flow. In the case of a stable motion, \( W \) is positive and the flow extracts energy from the blade.

Therefore, the system is stable if \( \delta_{\text{total}} \) is greater than zero. When \( \delta_{\text{total}} \) is nought, flutter is “achieved”. The phase shift can be a major parameter to visualize the areas of stability or instability. A positive phase shift does not necessarily translate into an unstable area. It depends on the scalar product of \( \vec{n} \) and \( \vec{e} \), and thus on the kind of motion (flexion for example).

b. Nodal diameter and chorochronicity

In a fan, even if the blades are supposed to be identical, they do not behave inevitably similarly. An inter-blade phase angle, which is caused by the coupling with the disk, can appear. This parameter may significantly affect the stability of the system since the surroundings of a given blade are different. Therefore, it is important to consider the coupling between blades and disk even if the computations are performed on one single blade. The inter-blade phase is narrowly linked to the notion of nodal diameter.

\[
\\sigma = \frac{2\pi n}{N_b}
\]  

(4)

Where \( \sigma \), \( n \) and \( N_b \) are the inter-blade phase angle, the nodal diameter and the number of blades, respectively.

The nodal diameter [5] is the number of lines in a fan that have the same motion at a given moment (Figure 2a). To take into consideration the nodal diameter during unsteady calculations, a condition of chorochronicity is set. A nodal diameter can be interpreted as two waves, one in advance and one in delay as seen on Figure 2b. Supposing that the flow is identical in each blade but at a different time, the two plans surrounding the studied blade will have the same boundary conditions, but shifted in time.

![Figure 2: Illustration of nodal diameter (a) and boundary conditions of chorochronicity (b) [1]](image)

**Numerical approach**

Calculations are performed using software 2, based on the elsA CFD solver written by the French Aerospace Lab ONERA.

a. Determination of flutter

The numerical method used to study flutter consists at first in performing a steady computation for a given rotational speed of the fan in order to determine the mean flow quantities, such as pressure. Once the steady state has been resolved, the motion of the blade is imposed, at a frequency corresponding to one of the natural frequency of the blade (as determined by preliminary mechanical computations).

The stationary study is mandatory to alleviate the problems of convergence sometimes observed during the unsteady calculations.
In our study, the modal amplitude is arbitrarily set to 1mm. The effect of the aerodynamic forces on the amplitude is not taken into account, which induces a simplification of our non-linear study, and a gain in computational speed. This simplification does not affect our study since the aim is to model the flutter’s onset. The only paramount parameters are the aerodynamic damping and the mass flow. If the total damping is equal to zero, the blade is subject to a flutter phenomenon.

If the total damping is negative, an interpolation from total damping and mass flows of other working points on the rotational speed line is performed to approximately estimate the mass flow for which the total damping is equal to zero. Once the mass flow is known, another interpolation from the steady results is performed to estimate the pressure ratio. There is one value of zero damping for each combination of rotational speed, blade mode and nodal diameter.

a. Computational set up

Given that the subject of interest is transonic flutter, the rotational speeds of the fan that are considered are respectively 90%, 94%, 96% and 100% of N (% of reference speed), which correspond to high rotational speeds.

Only the first mode of flexion (1F) and the nodal diameters comprised between 1 and 2 (1D or 2D) are considered. These choices are empirical and based on previous experiences. The test bed results show that the critical nodal diameters are 1 and 2 depending on the rotational speed. The natural frequency depends on the rotational speed, for the shape of the blade is supposed to change slightly with the rotation speed.

The compressible 3-D Reynolds averaged Navier-Stokes (RANS) equations for arbitrary moving bodies are solved using a cell centered finite-volume method. A Roe flux differencing scheme is used for the spatial discretization, while a backward Euler integration with implicit LU schemes [4] is applied for the temporal discretization.

For time accurate computations, the implicit dual time stepping (DTS) method [3] is employed. Whether for the steady or unsteady calculations, the turbulence modeling for RANS equations is the k-epsilon model.

The boundary conditions at the inlet (Figure 3) are conditions of injection. Four parameters out of five are set: the total pressure, the total temperature and the velocity directions (two angles). Regarding the outlet conditions, an association of radial equilibrium and valve law boundary conditions is used. The mass flow of the primary flow is imposed, while a numerical coefficient called valve boundary coefficient is applied to the secondary flow to reduce or increase pressure (and thus intake mass flow) in order to simulate, for example, transient regimes or engine wear and hence the rotational speed lines observed on figures 2 and 5.

The shroud, hub and nozzle are supposed to be adiabatic. This assumption is coherent since the heat transfers are negligible compared to work exchanges because of the short transient times of the flow through contact with solid boundaries.

![Figure 3: Meridional view of the fan (deformed geometry)](image)
Optimization methods

The influence of some parameters on the numerical results has to be studied in order to determine how the unsteady results can be improved.

a. Position of the inlet and outlet constant abscissa lines

The position of the inlet and outlet plans (Figure 3) used to generate the mesh may have an effect on the unsteady results, most notably on the inlet part because of its possible interaction with the pressure gradient generated by the leading edge shock wave.

Three new meshes, in addition to the initial mesh (IM), will be generated; one configuration with the inlet line shifted forward with respect to the leading edge (Fw), one with the outlet line shifted backward with respect to the trailing edge (Bw) and the last one with both the inlet and the outlet lines moved (Bw_Fw). Each plan is offset by a distance equal to one chord line.

The new meshes are generated using the Autogrid software of Numeca. A good mesh is required to achieve optimal computational conditions. Two conditions have to be fulfilled: the minimum skewness angle has to be superior to a certain value and the expansion ratio inferior to a maximum value.

b. Boundary conditions

It is a well known fact that in computational fluid dynamics, boundary conditions have a strong impact on the results. They dictate the particular solutions to be obtained. Choosing the wrong type of boundary conditions may result in problems of convergence or unphysical results. The impact of three different kinds of boundary conditions will be tested: at the inlet, conditions of injection (Inj) or non-reflection (Nref), and at the outlet, an association of radial equilibrium and valve law boundary conditions (VLaw) or conditions of non-reflection.

As opposed to the two other kinds of boundary conditions previously described, conditions of non-reflection consist in setting all the average physical parameters. Small variations around the average values are allowed. It aims at avoiding any reflection susceptible of disturbing the results.

c. Choice of the operating points

The different modifications will not be applied on each case. Only two cases are chosen for computational speed and efficiency reasons. To choose these cases, the cartography of the Mach number distribution of each case (as obtained from the steady calculations) is observed. The first one selected is the case for which the shockwave is the most spread out. The second case chosen will be based on its relevance regarding test results which corresponds to the case 94% N-1F2D.

Analysis of results obtained with Software 2 using Software 1 numerical methodology

The software 2 has been developed to replace the software 1. To check that it works well, a steady and unsteady study based on Software 1 numerical methodology is performed. As the two softwares use the same CFD code, the numerical results should be identical.

a. Steady Results

The compressor map is drawn from the steady computational results. The operating line of the compressor map (green curve on Figure 4) is a prescribed data. There are four rotational speeds studied (90%, 94%, 96%N and 100% of N) with software 2. The 96% of N was not computed with software 1.
Figure 4: Compressor Map with software 1 (blue) and 2 (red)

b. Cartography of Mach number

The shock wave at a height of 93% of the normalized radius is visualized. In our case, the relative Mach number is chosen to see the extent of the shock wave.

Figure 5: Cartography of the Mach number at a high valve coefficient for 4 rotational speeds (Blade to Blade view, 93%H) (a) and at 90% of N for 4 valve coefficients (Blade to Blade view, 93%H) (b)

c. Unsteady Results

The flutter margin calculated from the results of software 1 and 2 for the first mode (1F) and the first nodal (1D) diameter is plotted in Figure 6a. The aerodynamic damping for a rotational speed of 90% of N, a valve coefficient of +0.15 and a nodal diameter equal to 1 is depicted in Figure 6b.
The aerodynamic damping and the radial repartition of work are drawn for the case 90% of N-dp00-1F1D.

![Figure 6: Comparison of flutter margins (a) and Aerodynamic damping at 90% of N, high valve coefficient and for a nodal diameter of 1 (b)](image)

d. Discussions

The steady calculations are almost similar between software 1 and 2, except for low intake mass flows (high valve coefficients) as seen on Figure 4. For high valve coefficients, the pressure ratio decreases sharply for each rotational speed due to the separation of the boundary layer, as seen on Figure 5b. Complex phenomena such as stall are difficult to compute accurately, hence the slight differences observed between software 1 and 2.

The flutter margin (1F1D) between software 1 and 2 depicted in Figure 6a for different rotational speeds is slightly different, especially for very high rotational speeds. The maximal relative error of the case 1F1D is 7%. The total damping has to converge well to accurately estimate the mass flow associated with flutter onset. The unsteady results of software 2 converge well for low valve coefficients (Figure 7a) but often exhibit poor convergence for high valve coefficients, as seen in Figure 6b, and most of the time the total damping becomes negative for high valve coefficients. Given that the flutter is calculated from an interpolation of total damping and mass flows, a difference of unsteady results or approximate values of the aerodynamic damping induce a difference in flutter margins. Moreover, with software 2, and whatever the values of the valve coefficient, for the first periods, there is an overshoot of the aerodynamic damping that is not reproduced with software 1, as seen in Figure 6b. The flutter margin 1F2D gives close results to the flutter margin 1F1D, except for very high rotational speeds. The physical behavior of the results between softwares 1 and 2 are close, as seen in Figure 7a and Figure 7b. Be it on the pressure or suction sides, the radial repartition of work for software 1 and 2 are similar, except close to the tip (Figure 8b and Figure 8c).
Regarding the difference of flutter margins between computational and test bed results, the decrease of the flutter margin at 94\% of N compared to 90\% of N is not observed on the computational results (Figure 6a) since the flutter margin increases with the rotational speed, hence the necessity to improve the computational methodology.

From the visualization of the relative Mach number around the blade, as illustrated in Figure 5a and Figure 5b, it is observed that the higher the rotational speed, the higher the shock spreads. The valve coefficient has very slight effect on the shock unless there is a separation of the boundary layer. Therefore, one of the cases chosen is 100\% of N-dp15-1F1D. The other one is, as stated in the previous section, the critical case 94\% of N-dp10-1F2D according to test beds.

**Investigation of the modeling practices**

The current methodology seems improvable for the unsteady calculations in transonic areas since the engine-test flutter pocket is not well reproduced. Some numerical parameters are modified in order to see their effect on the results.

a. Effect of the boundary conditions and the positions of the inlet and outlet plans

The radial repartitions of aerodynamic work for the case 100\% of N-dp15-1F1D as a function of the boundary conditions are displayed in Figure 8 and Figure 9. The same operating point is considered. The curves only display results with conditions of injection and valve law boundary conditions and with conditions of non-reflection. The results with other conditions are reported in Appendix a.

![Figure 8: Radial repartition of aerodynamic work on the suction side (a) and pressure side (b)](image)

![Figure 9: Radial repartition of aerodynamic work on the suction side (a) and pressure side (b)](image)

The phase and the module of the unsteady pressure and the aerodynamic work, at a height of 95\% on the suction side and the pressure side and as a function of the abscissa lines at the inlet and the outlet and of the boundary conditions are depicted below, for the same operating point. Concerning the chosen convention, the pressure side varies between -1 (trailing edge) and 0 (leading edge), and the suction side ranges between 0 (leading edge) and 1 (trailing edge). The peak observed on the module of the unsteady pressure on the suction side at 0.3c (chord length) corresponds to the shock.
Figure 10: Module of the unsteady pressure for conditions of injection and valve law boundary conditions (a) and non-reflection conditions (b)

Figure 11: Phase for conditions of injection and valve law boundary conditions (a) and non-reflection conditions (b)

The aerodynamic damping of the case 100% of N-1F1D and the critical case 94% of N-1F2D as a function of the boundary conditions and the flutter margin with conditions of non-reflection and the initial mesh are drawn.

Figure 12: Aerodynamic damping for the cases 100% of N-1F1D (a) and 94% of N-1F2D (b) and flutter margin of the software 2 (c) as a function of the boundary conditions

Some values of aerodynamic damping of the case 94% of N-1F2D are not included in the Figure 12b due to problems of convergence of the unsteady calculations.

b. Discussions

For valve law boundary conditions (Figure 8 and Figure 13), the position of the inlet and the outlet plans has a strong effect on the radial repartitions of work. When the plan at the outlet is shifted backward, whatever the boundary
conditions at inlet, the suction side is stable and the pressure side is unstable. The phase and the module are also different as a function of the position of the mesh plans.

For non-reflecting boundary conditions at the inlet and at the outlet (Figure 9), the radial repartitions of work are close whatever the position of the plans. The phase is quite identical (Figure 11b), and the module of the unsteady pressure is very close, except around the shock (Figure 10b). The pressure peak corresponding to the shock (Figure 10b) is more or less important, depending on the position of the mesh plans. The observations that emerged from the study of the case 100% of N-dp15-1F1D and 94% of N-dp10-1F2D are similar.

The value of aerodynamic damping reinforces our observations. The standard deviation is the lowest for conditions of non-reflection (Figure 12a and Figure 12b). Moreover, the value of aerodynamic damping with conditions of non-reflection and the initial mesh is slightly lower than the one calculated with the current methodology for the critical case (Figure 12b). For the case 100% of N-1F1D (Figure 12a), it is higher, a tendency that does agree with the experimental results.

If the numerical model is well defined and the shock on the pressure side well-described, the unsteady results should not depend on the position of the plans at the inlet and at the outlet. The difference of unsteady results for conditions of injection or valve law boundary conditions is due to problems of reflection. Reflected waves disturb the results, which induces errors. The methodology using conditions of injection and valve law boundary conditions seems inaccurate. Applying conditions of non-reflection is more pertinent and does not disturb the results (Figure 12b).

Further to the previous study, the flutter margin 1F2D was drawn for conditions of non-reflection (Figure 12c) with non-improving results. The pocket of flutter around 96% of N found on test beds is not well reproduced. The flutter margin is close to the one found with the current methodology. Nevertheless, it is more precise because the aerodynamic damping converges well, even for high valve coefficients. There is no more “overshoot” (Figure 17) whereas the flutter margin calculated with the current methodology is imprecise due to problem of convergence for high valve coefficients.

Conclusion

The study aims at improving the modeling techniques of transonic flutter on a fan from a new software, called software 2. The current CFD tools fail to systematically reproduce the flutter pocket observed on some fans in transonic areas, around 96% of N in the case of our engine. The first part of the study consists in testing the new software, by comparing the steady and unsteady results to the ones of the previous software, named software 1. The same code and methodology are used. The results of the two softwares should be similar. Once done, the influence of some numerical parameters on the results has to be investigated in order to improve the current methodology. Only two cases are chosen to gain in speed and efficiency. If the results are improved or more relevant, a new study has to be performed to compare the new flutter margin to the one found on test beds.

The study shows that the steady results between the two softwares are similar. The unsteady results are also very close, except for high valve coefficients. The convergence of the aerodynamic damping, required to determine the flutter margin is longer, due to a high overshoot that does not appear with software 1. The flutter margin between the two softwares and the radial repartition of work are quite identical. Nevertheless, as expected, the pocket of flutter around 96% of N is not well reproduced.

The position of the mesh plans at the inlet and the outlet has a strong impact on the unsteady results such as aerodynamic damping, radial repartition of work, phase and module of unsteady pressure with the current methodology, whereas a well defined numerical model should not depend on this parameter. These differences are likely due to problems of reflections which disturb the solution. Using conditions of non-reflection seems more accurate since results depend very slightly on the positions of the plans. Only a few differences are observed on the peak of module of unsteady pressure around the shock on suction side. But the flutter margin with condition of non-reflection does not display a pocket of flutter around 96% of N. Nonetheless, with conditions of non-reflection, there is no more problem of overshoot and the convergence is quicker.

As a conclusion, the current methodology is improvable due to problems of reflected waves that may disturb the unsteady results. Using conditions of non-reflection prevents this problem from occurring but the new flutter margin
remains sometimes badly predicted. The investigation of other parameters such as a new temporal scheme (GEAR, Appendix b) or a finer mesh to better reproduce the peak of pressure around the shock may improve the numerical results.

Acknowledgment

I would like to acknowledge and extend my gratitude to the following persons who have helped me to achieve this project: Mr. SM, my local supervisor at Snecma who helped me join the team quickly, who spent time to answer all my questions and who actively participated in the success of this internship; Mr. Ulf Ringertz, my KTH examiner who gave me advice on the writing of the report, the Methods Department and the Design Office at Snecma who followed the progress of my work and proposed new guidelines for reflection, and Mr. MM, Head of the unit, and Mr. VP, Head of department, who monitored the project and corrected my report.

References


Appendix

Appendix a. Effect of the position of the abscissa lines and the boundary conditions

The radial repartitions of aerodynamic work for the case 100% of N-dp15-1F1D as a function of the boundary conditions are displayed for the same operating point (case 100% N of dp15-1F1D of initial mesh).

Figure 13: Radial repartition of aerodynamic work on the suction side (a) and pressure side (b) for conditions of non-reflection and valve law boundary

Figure 14: Radial repartition of aerodynamic work on the suction side (a) and pressure side (b) for conditions of injection and non-reflection

The phase, the module of the unsteady pressure and the aerodynamic work at a height of 95% on the suction side and the pressure side as a function of the abscissa lines at the inlet and the outlet and the boundary conditions are depicted below.
Figure 15: Variation of the phase for conditions of non-reflection and valve law boundary conditions (a) and for conditions of injection and non-reflection (b) on the suction side and the pressure side.

Figure 16: Variation of the module for conditions of non-reflection and valve law boundary conditions (a) and for conditions of injection and non-reflection (b) on the suction side and the pressure side.

The curves of convergence of aerodynamic damping as a function of the boundary conditions are drawn.

Figure 17: Aerodynamic damping as a function of period for different boundary conditions for the case 100% of N-dp15-1F1D.
Appendix b. Dual time stepping

The URANS equations cannot be solved directly by an implicit time scheme because the unsteady solutions depend on time.

A first option is to use an explicit scheme, from the “Unsteady” method proposed on the software 2 but it has drawbacks. Is it an explicit one-order accuracy scheme and the time step chosen has to be low enough to respect the CFL condition. If the time scales of the physical problem are much higher than the ones derived from the CFL condition, the explicit method will require a time step lower than necessary, due to the Courant number, what can be considered as a waste of time.

A second option is to add a fictive time step, and to transform an unsteady problem into a pseudo-steady problem to be able to use an implicit method [3]. The implicit method does not depend on the CFL condition, and therefore higher time steps may be used. Moreover, it is a two-order accuracy scheme. There are two time steps: the physical one and the fictive one. For each physical iteration, the unsteady solution is found by performing steady iterations to make the field converge.

Two different methods uses a fictive time step: the DTS (Dual Time Stepping) [3] and the GEAR. The difference lies in the definition of the fictive time step for each physical time step:

- The GEAR uses a global time step hence a coherent convergence in the whole field. The velocity of information propagation is the same everywhere.
- The DTS uses a time step depending on the size of each cell hence a different convergence in the whole field.

Figure 18: Discrétisation temporelle avec pas de temps dual