Fluid Structure Interaction based Flutter Prediction

A. A. Belkhede[1], Dr. K. S. Zakiuddin[2], P. B. Khope[3]

#Department of Mechanical Engineering, Priyadarshini College of Engineering, RTM Nagpur University, India

[1] akashbelkhede@live.com

[2] qszaki1@rediffmail.com

[3] khopepb@yahoo.co.in

***Abstract* — Focus of this paper is to study Flutter based on coupled Fluid – Structure interaction. This paper includes detail discussion on flutter, design of blade of an axial compressor.**

***Keywords*— Flutter, Fluid Structured interaction, fans and compressor, blade design.**

I. Introduction

The design trends for modern commercial aircraft engines are towards high thrust to weight ratio, fuel efficiency and safety. These requirements lead to design fans and compressor with high pressure ratio per stage, thin blade, making blades more susceptible to fluid induced vibration.

The engineering science of aeroelasticity has been described by Collar (1946) as the study of the mutual interaction of the inertial, elastic and aerodynamic forces on structural members exposed to an airstream and the influence on design [1]. Turbomachines ruled by compressible flows are designed to create a great pressure differential. The most sounding example is the gas turbine cycle used in airplanes, automobiles and even boats: air flows through compressors, combustors and turbines. The latter extracts some if not all energy, depending on the applications, i.e. a fighter doesn't need to pass much energy to a shaft while in a Thermoelectric, that is the main purpose. Although there are considerably distinct applications, the flow structures are similar between all and it becomes important to understand how they work in detail. Computational Fluid Dynamics (CFD) and the advance in computing performance in the last decades allowed complex and large simulations to be made in reasonable time, thus much of the pre-project budget is reduced. Furthermore, it has proved to be a powerful research tool when examining complex physics and multi-phenomena interaction, namely on turbomachinery [1].

II. Problem in Present Design

The design trends for modern commercial aircraft engine are towards high ‘thrust to weight’ ratio, fuel efficiency and safety. These requirements lead to design fans and compressors with high pressure ratio per stage, thin blades, making the blades more susceptible to fluid induced vibration.

III. Literature Review

A. Generalised modelling

The governing equations in integral form are written for a dynamic deforming grid,

$$\frac{d}{dt}\int_{\Omega(t)} q\, d\Omega + \oint_{S(t)} (F_c - F_v)\cdot \hat{n} dS = \int_{\Omega(t)} H d\Omega$$

where **q** is a conservative variable vector, $\Omega(t)$ and $S(t)$ are control volume and its boundary, **n** is the out-pointing norm vector of $S(t)$, $\boldsymbol{F_c}$ and $\boldsymbol{F_v}$ represent the convective and viscous fluxes, and ***H*** is the source term vector consisting of non-inertia force due to constant rotation of the system and other external contribution to viscous fluxes is modelled with several eddy-viscosity models.

There are a number of more refined and realistic structural models of the blade vibration under aerodynamic loads. However, in the context of coupled fluid-structure solution, such model generally requires expensive computing resource which is not currently available. Linear aeroelasticity model is proven to be sufficient for most flutter problems. The structural dynamic equation for the blade vibration is written as

$$M\ddot{x} + C\dot{x} + kx = Q$$

Where ***M*** is the mass matrix, ***C*** the damping matrix, ***K*** the stiffness matrix, ***x*** the displacement vector and ***Q*** the force vector.

The structural equation of motion is a set of linear differential equations that can be solved with a modal approach in Eigen-space. Once the Eigen-solution is obtained, the vibration of structure can be expressed as a series of linearly independent modal vectors, so the deflection of blade can be written as a linear combination of the modes included in the analysis

B. Basics of blade Geometry.

Following is the generic blade geometry with explanation of NACA/NASA terminology.

Leading Edge: It is an edge at the front of rotor.
Trailing Edge: It is an edge at the rear of the rotor.
Thickness: It is the perpendicular distance over the chord line.

Camber Line: The camber line that is commonly divided between upper and lower line of rotor, and also called mean camber line.
Camber: Camber is the normal distance between chord line and camber line.
The constants like Z_c, Z_t, Z_u, Z_l can be calculated by following formulas.

1. $Z_c=(Z_u+Z_l)/2$
2. $Z_t=(Z_u-Z_l)/2$
3. $Z_u=(Z_c+Z_t)/2$
4. $Z_l=(Z_c-Z_t)/2$

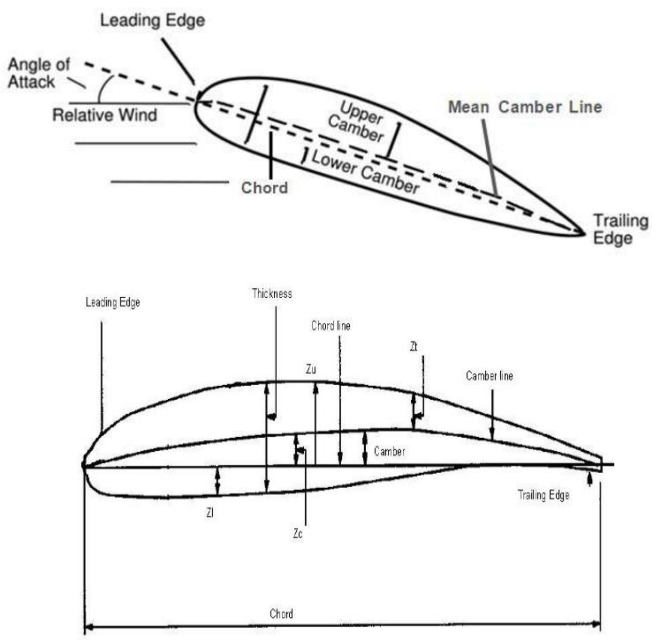

Fig 1. Schematic Diagram of Blade

C. Overview of theory used in mathematical modelling

The first assumption is that the flow over the airfoil must be moving through the fluid at subsonic speeds. This is important because at speed approaching the speed of sound, shockwave occurs in which the fluid flow no longer becomes continuous, and the perfect fluid idealization beaks down.

The second assumption is that the flow over the airfoil satisfies kutta condition. The kutta condition states that the fluid flowing over the upper and lower surface of airfoil meets at trailing edge of airfoil.

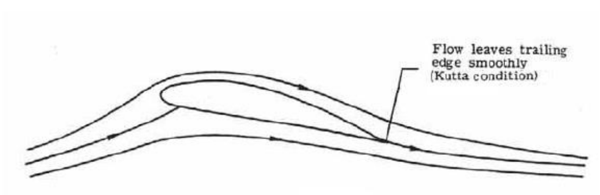

Fig. 2 Kutta condition applied

IV. Proposed Solution

The complete design process for the compressor will encompass the following steps.

1. Choice of rotational speed and annulus dimension.
2. Determination of stages based on assumed efficiency.
3. Calculation of air angles at every stage at the mean radius.
4. Determination of air angle for each stage from root to tip.
5. Investigation of compressibility effect.
6. Selection of compressor blading using cascade data.
7. Check on efficiency previously assumed, using the cascade data.
8. Estimation of off-design performance.
9. Rig testing.

Being an academic project steps 8 and 9 will be skipped.

After following these steps we will get the physical dimensions of blade such as chord length, air inlet angle, air outlet angle, number of blades at each stage, blade height etc.

A CAD model is then prepared using the data obtained from the above design consideration, then the same is then imported to CFD analysis software, ANSYS to investigate whether proposed design fulfilled the required condition.

CFD results are then exported to FEA software to investigate flutter, in which mode shapes and natural frequencies will be calculated using CREO-Mechanica, FEA tool of Pro-E.

V. Conclusions

Flutter phenomena has been widely studied for standard airfoil available such as NASA 67 rotor, NASA 37 rotor. As attempt is made to design a separate airfoil, for the flutter prediction. The proposed research is intend to drive following results,

- Mathematical design of blades of an axial compressor
- CFD analysis of the blade for flow characteristics
- FEA analysis for Mode shapes, and Natural frequency obtained due to strong fluid interaction.

Coupled fluid-structure method is not only capable of predicting the stability of the blade by analysis of the vibration history, but also able to compute the amplitude and frequency of the vibration, which is important to evaluate the dynamic stress for reliability and life analysis.

REFERENCES

[1] Chen-an Zhang[1], Zhengyin Ye and Feng Liu, Numeric research on aeroelastic problem of a rotor due to IGV/Fan interaction, 47th AIAA Aerospace Science Meeting, Jan 2009.
[2] ZHENG Yun, YANG Hui, Coupled Fluid-structure Flutter Analysis of a Transonic Fan, Chinese Journal of Aeronautics 24 (2011) 258-264.
[3] Maria Angelica Mayorca, Development and Validation of Numeric tool for the Aeromechanical Design of Turbomachinery, Department of Energy Technology, School of Industrial Technology and Management, Royal Institute of Technology, Stockholm, Sweeden, 2010
[4] HIH Sarvanamutto, H. Cogen, GFC Rogers, "Gas Turbine Theory", Pearson publication, fifth edition 2012, ISBN: 978-81-7758-902-3