ISSN : 2248-9622, Vol. 5, Issue 5, (Part -6) May 2015, pp.149-154

RESEARCH ARTICLE OPEN ACCESS

Fluid Structural Modal Coupled Numerical Investigation of Transonic Fluttering Of Axial Flow Compressor Blades

Rio Melvin Aro.T*, EZHILMARAN G.**
*(Department of Aeronautical Engineering, Anna University, Chennai.)
** (Department of Aeronautical Engineering, Anna University, Chennai.)

ABSTRACT
Flutter is an unstable oscillation which can lead to destruction. Flutter can occur on fixed surfaces, such as blades, wing or the stabilizer. By self-excited aeroelastic instability, flutter can lead to mechanical or structural failure of aircraft engine blades. The modern engines have been designed with increased pressure ratio and reduced weight in order to improve aerodynamic efficiency, resulting in severe aeroelastic problems. Particularly flutter in axial compressors with transonic flow can be characterized by a number of aerodynamic nonlinear effects such as shock boundary layer interaction, rotating stall, and tip vortex instability. Rotating blades operating under high centrifugal forces may also encounter structural nonlinearities due to friction damping and large deformations. In the future work a standard axial flow compressor blade will be taken for analysis, both Subsonic and Transonic range are taken for analysis. Fluid and Structure are two different domains which will be coupled by full system coupling technique to predict the fluttering effect on the compressor blade. ANSYS is a commercial simulation tool, which will be deployed in this work to perform FSI (Fluid Structure Interaction) and FSI coupled Modal to predict the flutter in the compressor blades
Keywords – Flutter, Compressor Blade, FSI Modal Analysis, ANSYS

I. Introduction

1.1 Problem Description

The most difficult and computationally expensive component of flutter prediction is the determination of unsteady aerodynamic forces. Flutter usually occurs at off-design operating conditions where flow separation occurs, for example, stall flutter and choke flutter. 3D Coupled Fluid-Structural flow modelling is necessary to accurately predict separated flow. In the past, 3D unsteady viscous flow modelling has been too computationally expensive to use during the design phase. Manufacturers have had to rely on simplified flow models (e.g. 2D inviscid), reduced order models and empirical data during the design process. It has been shown that simplified models and empirical data fail to capture the physics of unsteady flow, and hence have a limited range of applicability. So, it is evident to have a full scale 3D fluid structural modal transient simulation technique to predict the flutter in compressor blades preciously

1.2 Proposed Solution

To predict the fluttering effect on axial compressor blades a fully coupled fluid-structural-modal simulation method is proposed. ANSYS CFD tool Fluent and Structural tools are effectively coupled with Structural-Modal with strong system coupling method. By performing this simulation, the blade flutter can be found and further damping and structural improvement suggested for blade design.

II. MODEL AND MODAL ANALYSIS

2.1 Vibration

Free vibration takes place when a system oscillates under the action of forces inherent in the system itself due to initial disturbance, and when the externally applied forces are absent. The system under free vibration will vibrate at one or more of its natural frequencies, which are properties of the dynamical system, established by its mass and stiffness distribution. In case of continuous system the properties of the system are the function of spatial coordinates. The system has infinite number of degrees of freedom and infinite number of natural frequencies. In actual practice there is always some damping (e.g., the internal molecular friction, viscous damping, aerodynamical damping, etc.) present in the system which causes the gradual dissipation of vibration energy, and it results gradual decay of amplitude of the free vibration. Damping has very little effect on natural frequency of the system, and hence, the calculations for natural frequencies are generally made on the basis of no damping.

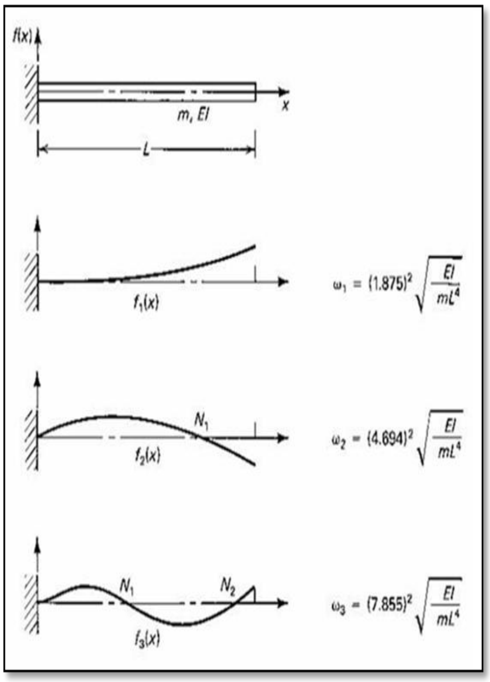

Fig.2.1- Cantilever Beam Undamped Mode Shape and Frequency

Damping is of great importance in limiting the amplitude of oscillation at resonance. The relative displacement configuration of the vibrating system for a particular natural frequency is known as the Eigen function in continuous system. The mode shape corresponding to lowest natural frequency (i.e. the fundamental natural frequency) is called as the fundamental (or the first) mode. The displacements at some points may be zero. These points are known as nodes. Generally *n*th mode has (*n*-1) nodes (excluding end points). The mode shape changes for different boundary conditions of a beam. Rotor Blade is a cantilever Beam, so it is vital to understand the frequency and Mode shape for Analytical verification.

2.2 Mode Analysis Details

The following models show the modal analysis steps.

- Geometric Model (Rotor) Import into ANSYS-Structural
- Meshing of Compressor (Rotor & Stator)
- Applying Boundary Conditions (Fixing Root to Hub))
- Applying Material Property
- Performing MODAL Analysis
- Performing Post Processing to get Resonance frequency

2.3 Modal Analysis

Modal analysis is the study of the dynamic properties of structures under vibration excitation. Modal analysis is the field of measuring and analysing the dynamic response of structures and or fluids when during excitation. Modal analysis is the field of measuring and analysing the dynamic response of structures and or fluids when during excitation. Examples would include measuring the vibration of a car's body when it is attached to an electromagnetic shaker, or the noise pattern in a room when excited by a loudspeaker. Modern day modal analysis systems are composed of 1)sensors such as transducers (typically accelerometers, load cells), or non contact via a Laser vibrometer, or stereo photogrammetric cameras 2) data acquisition system and an analogy-to-digital converter frontend (to digitize analogy instrumentation signals) and 3) host PC (personal computer) to view the data and analyze it.

2.4 Analysis Configuration

Analysis Type	=	Modal
FE Model Type	=	Cyclic Symmetry
Mesh	=	3D – Tetra
Tip diameter	=	514 / 485 mm
Hub diameter	=	192.75/ 231.83 mm

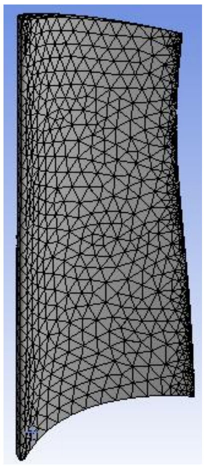

Fig.2.2 – Mesh Details

The following details are the mesh details of the model.

No. of Nodes	=	143279
No.of Elements	=	79636
Type of Element	=	SOLID187

2.5 Analysis Results

The following figures show the analysis results

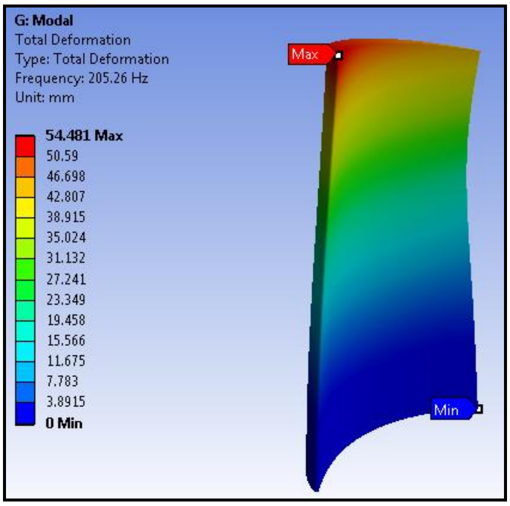

Fig.2.3 – First Mode Shape

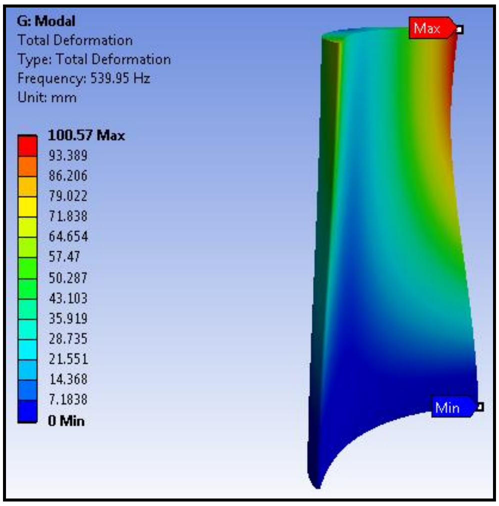

Fig.2.4 – Second Mode Shape

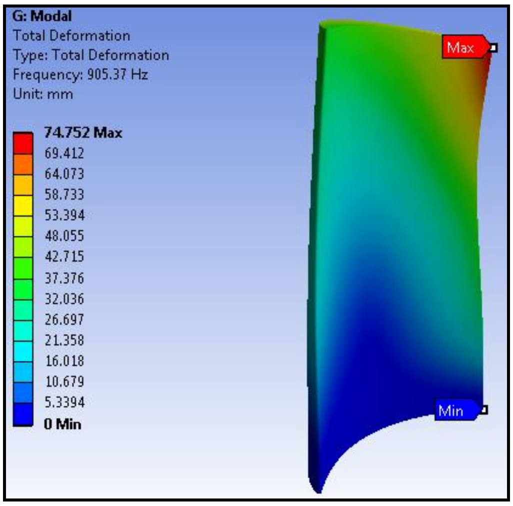

Fig.2.5 – Third Mode Shape

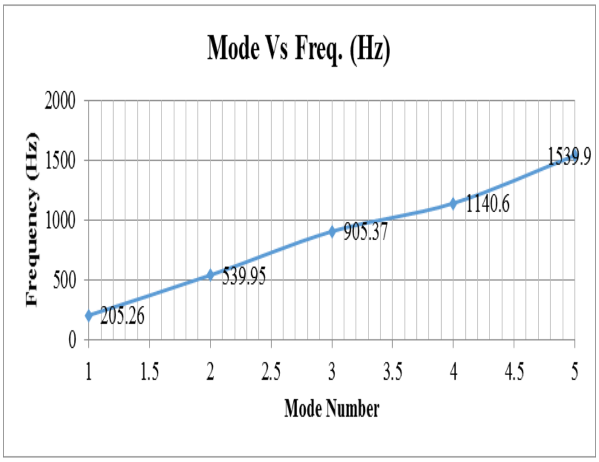

Chart 2.1 – Mode Shape Vs Frequency

2.6 Flutter Analysis

Prediction of flutter can be done through flow integrated vibration analysis; it is a coupled field simulation where the flow in the compressor is predicted through flow simulation and the results will be translated to structural analysis and then to modal analysis to get flutter of blades. The following figure shows the work flow involved in the flutter analysis.

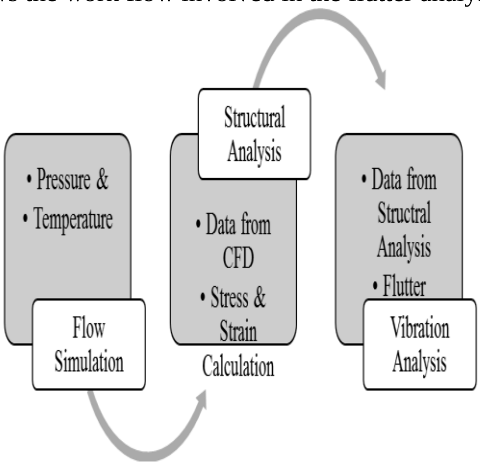

Fig. 2.6 – Work Flow- Flutter Analysis

The flow simulation (CFD) estimates pressure and temperature induced in the blade at design point and in chock and the data is then transferred to structural analysis to find out stress and deformation, which is then transferred to vibration analysis to predict flutter in blades. The following figures shows the results of the design point flutter

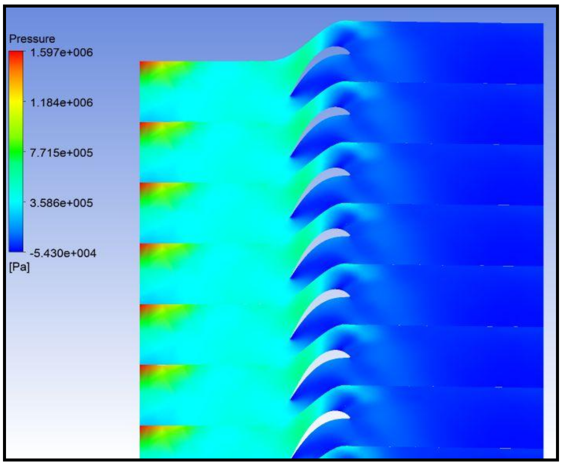

Fig.2.7 – Pressure – Blade to Blade View

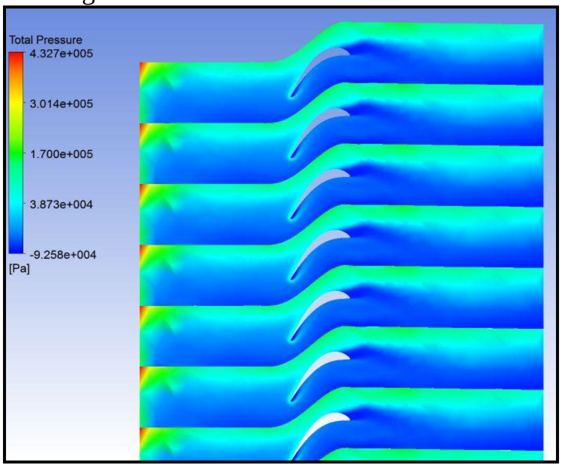

Fig 2.8 – Total Pressure – Blade to Blade View

The above blade to blade view shows the pressure and total pressure of the design point flow analysis.

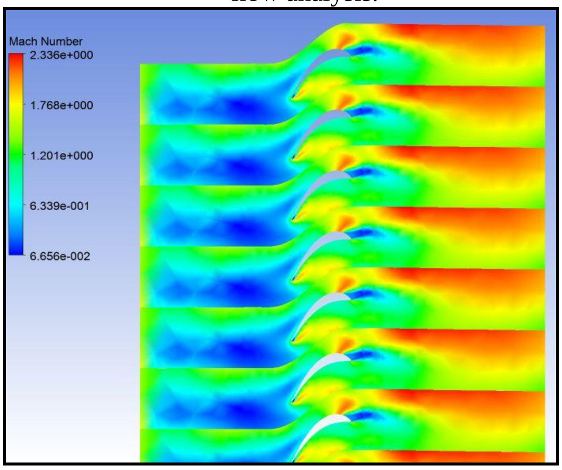

Fig. 2.9 – Mach Plot – Blade to Blade View

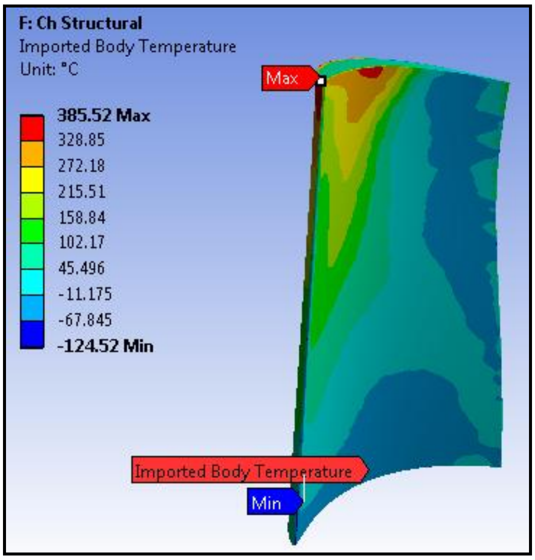

Fig. 2.10 – Imported Pressure and Temperature

The fig.2.10 shows the imported temperature and pressure load from flow analysis, theses analysis forms fluid structural interaction model which then transfers its results to modal analysis which produces the flutter results. The structural results show the induced stress in the blade due to flutter and its corresponding deformation and other relevant results quantities. The modal analysis followed by the FSI predicts the vibration due to flow pressure or flutter frequencies.

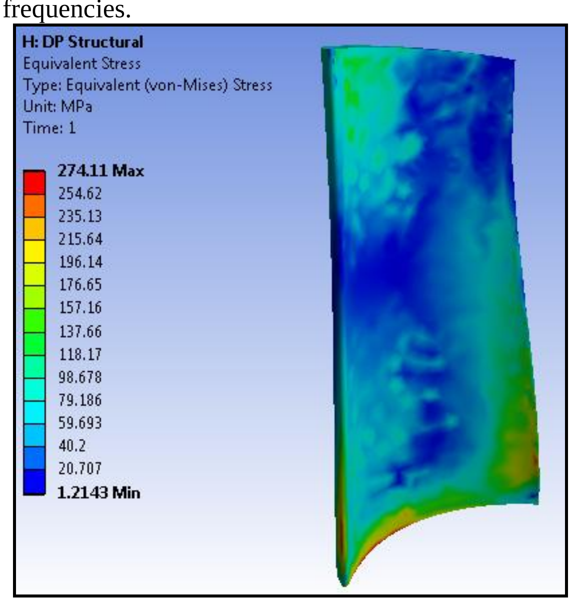

Fig.2.11 - vonMises Stress plot

The induced vonMises for design point is 274Mpa.

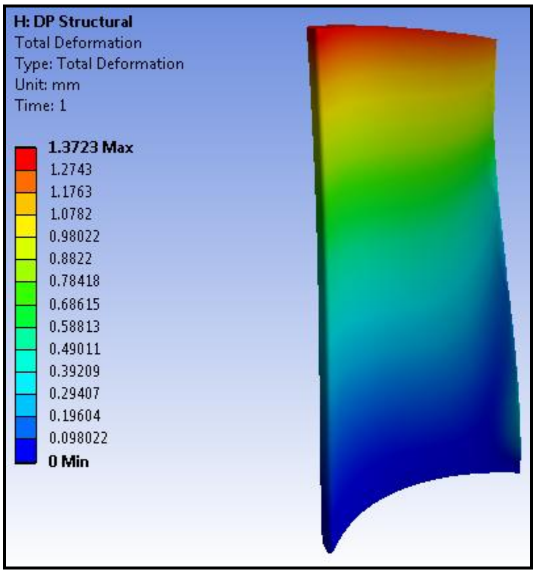

Fig. 2.12 – Deformation Plot

The above figure shows the deformation of the blade due to fluid structural interaction.

The following figures shows the flutter due to design point load

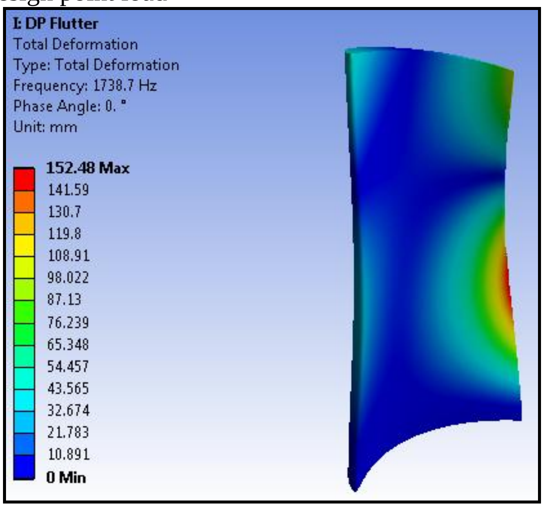

Fig. 2.13 – Design Point Flutter Results

It is clear that the design point flutter frequencies does not match with resonance frequency and ensures the safety of the flutter induction.

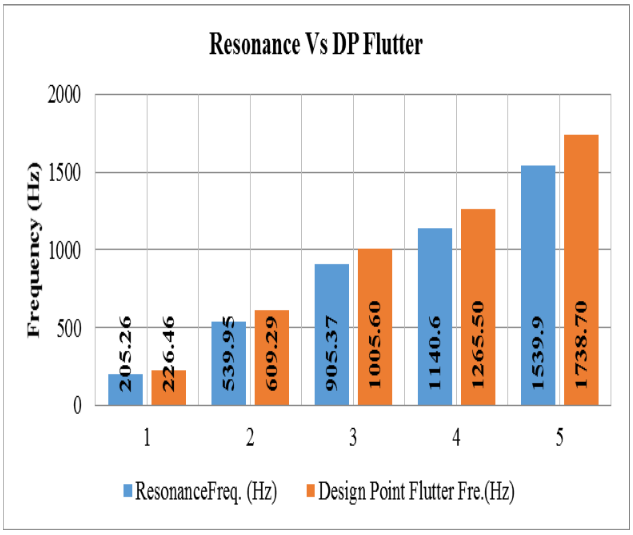

Chart 2.2 – Resonance vs Flutter (DP)

III. CONCLUSION

The initial phase of the work concluded with objective formulation followed by detailed literature review, where the fundamentals of axial flow compressor design is gathered, many reviews provided the design of NACA 67 rotor model, which is a standard model for studying flutter and is recommended for flutter study, many reviews provides flutter predication and its nature of occurrence and problem associated with it.

Problem identification is done and a proposed solution is formulated to predict the flutter in NACA 67 rotor model, which is the base model for all type of transonic compressor. A methodology is formulated for this projects work and the same is carried out successfully to predict the flutter at design and chock point.

The simulation is a tactical model for performing flutter predication in compressor blade, initially modal analysis is performed to find out the resonance frequency and then the flutter for design point mass flow and chock point mass flow is performed using flow combined structural and modal analysis system where the results of the flow simulation is transferred in the form of pressure and temperature to structural analysis and then its results are passed to modal to perform flutter predication

The results showed that flutter causes failure in chock point and safe with design point. This research can be further extended for experimental investigation which can give further insight into this flutter with different flight condition.

REFERENCES

[1] Cai X. et. al (2008) '*Performance of WENO Scheme in Generalized Curvilinear Coordinate Systems*', 46th AIAA Aerospace Sciences Meeting and Exhibit, American Institute of Aeronautics and Astronautics.

[2] Couch M.A. et. al (2004) '*A Flutter Model for Rotor Blades with Flaps'*, 24th International Congress of the Aeronautical Sciences.

[3] Gansel P.P. et. al (2012) '*Numerical Simulation of Low-Speed Stall And Analysis of Turbulent Wake Spectra'*, Conference on Modelling Fluid Flow, The 15th International Conference on Fluid Flow Technologies.

[4] Gourdain N. et. al (2007) '*Simulation of Rotating Stall in a Whole Stage of an Axial Compressor*', IJERT, Vol 1, No. 3, pg: 23-29.

[5] Hah C. et. al (2004) '*Role of Tip-Leakage Vortices and Passage Shock in Stall Inception in A Swept Transonic Compressor Rotor*', Proceedings of ASME Turbo Expo 2004.

[6] IM H.S. et. al (2012) '*Prediction of a Supersonic Wing Flutter Boundary Using a High Fidelity Detached Eddy Simulation'*, 50th AIAA Aerospace meeting.

[7] IM H.S. et. al (2011) *'Delayed Detached Eddy Simulation of a Stall Flow Over NACA0012 Airfoil Using High Order Schemes*', 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition.

[8] IM H.S. et. al (2013) '*Flutter Prediction of A Transonic Fan With Travelling Wave Using Fully Coupled Fluid/Structure Interaction'*, Proceedings of ASME Turbo Expo 2013: Turbine Technical Conference and Exposition.

[9] Lobitz D.W. et. al (2007) '*Flutter Speed Predictions for MW-Sized Wind Turbine Blades*', United States Department of Energy.

[10] Martinat G. et. al (2008) '*Turbulence modelling of the flow past a pitching NACA0012 airfoil at 105 and 106 Reynolds numbers'*, Journal of Fluids and Structures, Vol 24, pg: 1294–1303.