CHAPTER 3

STRAIN GAGE TEST DATA EXTRACTION

Experimental data has to be extracted from a Strain Gage Test (SGT) so that HCF (High Cycle Fatigue) lifing can be performed on the required blade. In addition, in this research project, the experimental data has to be extracted so that it can be correlated with the analytical results. Furthermore, to obtain the vibratory stresses analytically, the damping value has to be extracted from the data reduction since it cannot, currently, be calculated analytically. The data reduction will be done only on the high-pressure turbine blade for turbofans engine and the compressor turbine blade for the turboprops engines. The data reduction will be performed on the high-pressure turbine blade of the PWC Engine 1 and PWC Engine 2 engines, and on the compressor turbine blade of the PWC Engine 3 engine. These blades are uncooled, unshrouded (no inter-connection between blades) and the data extrapolated are only to be used on blades with the same characteristics for design. Furthermore, a damping value trend will be extrapolated as function of mode, natural frequency and harmonic of excitation. It is important to note that only the modes that are in resonance or close to will be of importance and studied.

3.1 Resonance Identification

A resonance is defined as a coincidence between a natural frequency of a component and a periodic excitation on a waterfall (Figure 4). A waterfall is obtained from a Fast Fourier Transform of a time signal given by a strain gage during an engine testing. The three axes represent the following: the frequency range, the engine rotational speed and the amplitude of the vibratory strain. Every horizontal line represents one capture engine speed during the test. During the test, the strain gage captures the vibratory strain at the determine location on a blade. The blade exhibits vibratory strain due to its own natural frequencies (almost parallel to the engine rotational speed axis) or due to the excitation sources. If the excitation is an integer of the rotation speed, such as upstream vane wakes, a diagonal line will appear on the waterfall. If the excitation line and the natural frequency line meet, a resonance will occur which is usually demonstrated by high amplitude vibratory strain.

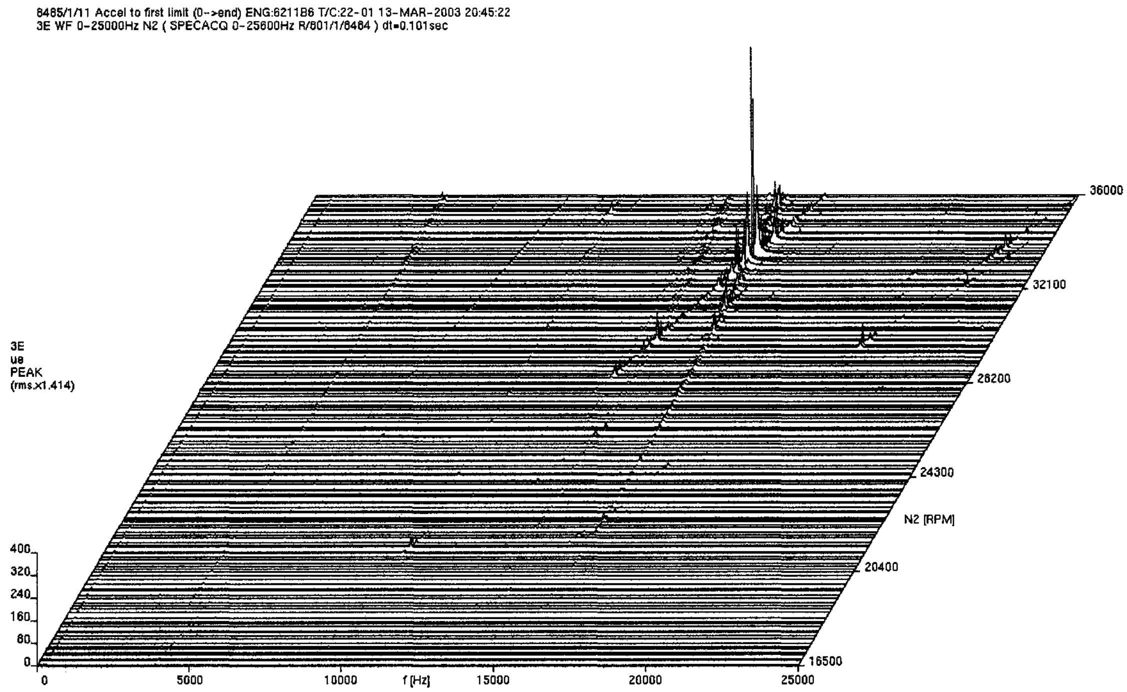

Figure 4 Waterfall 0-25000 Hz

In a turbine engine, periodic excitations can be generated by multiple components due to the rotating nature of the engine. Therefore, when designing a turbine blade, great care must be taken to the periodic excitations, mostly the vane passing, so that no resonance occur in the running range of the engine. When performing data reduction, resonance can be identified by the high amplitude of the strain compared to the rest of the frequencies and rotating speeds (Figure 4). Multiple resonances can be identified on each mode at specific natural frequency due to the different surrounding components and the excitation harmonics. The results are presented in section 7.1.1.

3.2 Modal Damping Extraction

When turbine blade vibratory stresses are predicted analytically, the total damping acting on the blade is critical to the accuracy of the stress value. The total damping is the sum of the aerodynamic damping and the mechanical damping. Furthermore, the mechanical damping comprises the structural damping of the material and the friction damping generated in the fixing area. These quantities depend on the CF load, metal temperature, frequency, material and surface finish at the blade-disc interface. The aerodynamic damping can be obtained by analytical calculations using Euler equations. As for the remaining damping values, the only accurate method to obtain them is from experimental testing. Therefore, from SGT data reduction, the total damping values have to be extracted so that the stress values can be predicted with accuracy.

The basic method to determine the total damping value is the following:

1- Identification of the resonance
2- Identification of the Engine Order range
3- Export the data
4- Curve fitting
5- Analytical tool

The identification of the resonance was presented previously (Figure 4). The goal of the Engine Order Plot is to build the blade response versus the rotor speed on an engine order excitation in order to have almost a constant force over a small rotor speed range. The Engine Order band has to be wide enough to capture the several adjacent spectral components that define the full peak. To make sure that the Engine Order Plot range is wide enough, a spectrum plot at the resonance speed is plotted from the waterfall (Figure 5).

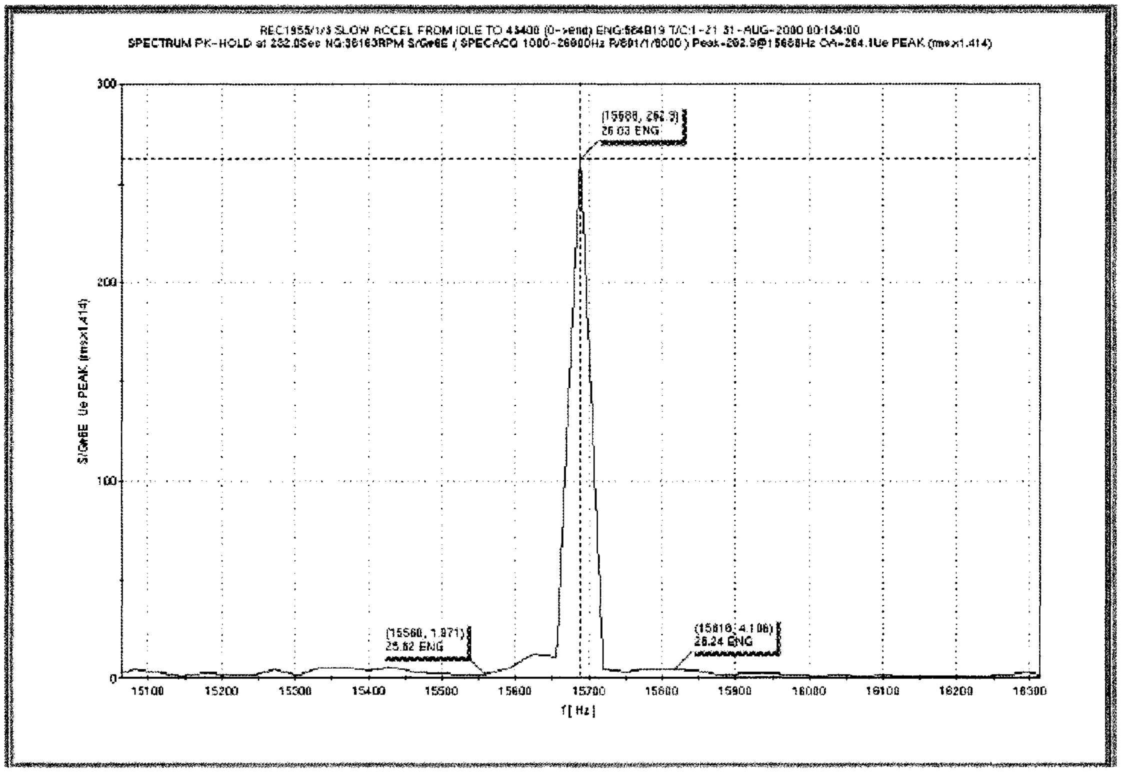

Figure 5 Resonance spectrum plot

Based on the spectrum plot, the Engine Order band is defined by incorporating the whole peak. The Engine Order Plot is relative to the engine rotor speed. To get the real resonant peak, an average over a frequency bandwidth for each engine revolution is done. To avoid any loss of information, the FFT frequency bandwidth has to be centered on the resonance frequency and has to be equal or less than 6400 Hz due to the limitation in resolution of the analyzer. Using this information, the data can be exported to an ASCII file for post-processing.

In order to extract the damping from the Engine Order curve, a single degree of freedom (SDOF) curve fitting method is used. This method is based on the viscous damping theory. As this is used locally (resonance), this is an acceptable assumption since, based on the experimental and analytical data, if the modes are uncoupled. The response of a SDOF in the rotor speed domain is:

$$X_i = \frac{D}{\sqrt{\left(1-\left(\frac{N_i}{N_c}\right)^2\right)^2 + \left(2\zeta\frac{N_i}{N_C}\right)^2}} \tag{3.1}$$

D	Equivalent static stress or strain (static deflection)
ξ	Damping ratio
N_c	Resonance speed (RPM)
N_i	Rotor speed (RPM)
X_i	Response of the component (stress or strain)

As the rotor speed range is known (N_i), the function X_i will be fully defined when D, N_c and ζ are known. The aim of the curve is to find D, N_c and ζ that best define a SDOF fit for the SGT data [15]. The least square method is used to achieve this.

The least square function is defined as:

$$\Pi(\zeta, N_c, D) = \sum_i (Y_i - X_i)^2 \tag{3.2}$$

With: X_i = theoretical SDOF curve as defined above over the rotor speed range
Y_i = SGT data over the rotor speed range
i = index that varies to dwell the rotor speed range of interest.

The parameters D, N_c and ζ that minimise the least square function will define the function X_i that best fit the SGT data Y_i. The damping factor ζ that is found with this method is assumed to represent the experimental damping.

A developed MATLAB® routine (APPENDIX 1) is used to fit the SDOF curve on the SGT data. This routine uses a MATLAB iterative solution. This function finds the parameters D, N_c and ζ that minimises the least square function. It also allows the user to specify the speed range of data that have to be used in the least square function calculation. The inputs of the routine are:

Table I

MATLAB® routine inputs

MATLAB variable name	Example	Description
sgt_data	'EO_REC1955_3_6F_2H.CSV'	The .csv file that was exported from the waterfall. It contains the Engine Order data versus the rotor speed. Some percent signs have to be added at the beginning of the text lines to comment them.
Nmin	34100	Minimum speed of the range of interest.
Nmax	38900	Maximum speed of the range of interest.
Tol_damping	1E-6	Termination tolerance of the damping factor.
Tol_reso_speed	1E-6	Termination tolerance of the resonance speed.
Tol_defl	1E-6	Termination tolerance of the static deflection.
Max_iter	1E50	Maximum number of iteration.
Dratio_init	1E-3	Initial value of the damping factor.
Delta_init	1.5	Initial value of the static deflection.

The results of the MATLAB® routine present the SGT data curve superimposed with the calculated SDOF analytical curve (Figure 6).

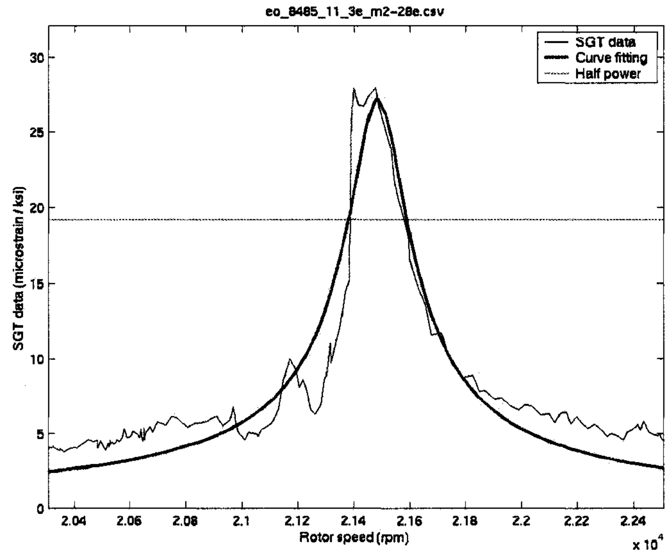

Figure 6 Curve fitting for damping extraction

The total damping value is given in the logarithmic decrement form. This method invokes that the forcing value throughout the resonance is constant (D parameter is constant). In reality, the force changes with regards to the engine rotating speed. Since the resonance band is very narrow ($\approx$ 1000 RPM), the change in the forcing value is deemed negligible. The results are presented in section 7.1.2.

3.3 Vibratory Stress Calculation

The experimental stress values are determined using the Hooke's Law:

$$\{\sigma\} = \{E\} \{\varepsilon\} \tag{3.3}$$

The deformation or strain (ε) is obtained from the data reduction plots. Since it is assumed that the highest stress value will be at the surface of the turbine blade, only one strain gage (one direction) is necessary to determine the vibratory stress. The Young's modulus (E) is dependent on three parameters. The first parameter is the type of material used for the turbine blade since different materials have different Young's modulus. The second parameter is the metal temperature of the blade at the location of the strain gage position. The Young's modulus decreases with the temperature elevation and therefore accurate temperature values are needed. The third parameter is the orientation of the strain gage. The materials used in a turbine blade are generally single crystal orthotropic materials. The orthotropic characteristic suggests that the Young’s modulus of the material is not equal in all the axes of the crystal. Therefore, the orientation of the strain gage must be taken into account to determine the correct Young's modulus value. The results are presented in section 7.1.3.

CHAPTER 4

FINITE ELEMENT MODEL BOUNDARY CONDITIONS DEFINITION

This subject will be concentrated on the application of boundary conditions using contact elements to determine the dynamic properties of a turbomachinery blade in an environment where the friction phenomenon is present. This study is performed using ANSYS® contact elements [18], which are meshed on the entire fixing area to simulate the interaction between the blade and the disc [16]. No assumptions were made initially for the blade-disc contact surface. The ANSYS® contact elements used require input for the static and dynamic friction coefficients and other parameters that have an effect on the convergence of the model. Before the modal analysis, a non-linear static analysis is performed with pre-stress effects, i.e. the blade metal temperature and the turbine shaft rotational speed. This static analysis, which is non-linear due to the addition of contact elements, calculates the new equilibrium position of the blade with respect to the disc due to the pre-stress effects. With the new equilibrium position found, a linear modal analysis is performed in order to obtain the natural frequencies and mode shape of the analyzed blade. The first four (4) natural frequencies and mode shapes are evaluated in this study. A convergence study is also performed to determine which contact element parameters have a significant influence on the natural frequencies values. The experimental results are extracted from strain gage tests for the natural frequencies and from a laser scan tests for the mode shapes. The analytical results are compared to the experimental results. Furthermore, experimental testing will be performed to determine the correct friction coefficient values as well as mode shape determination.

Contact elements are primarily used to simulate the contact stress and displacement between two moving components relative to each other. Current turbomachinery blade modal analyses are performed in PWC without the mating disc. The new method will include part of the disc, and the contact elements will be used between the blade and the disc fixing.

4.1 Current Analysis

The current analysis omits the displacement between the blade and the disc and assumes no motion of the blade. The turbine blade is meshed using tetrahedral 10-node parabolic elements (SOLID92). These analyses typically have approximately 40,000 elements, 60,000 nodes and 180,000 degrees of freedom. The boundary conditions consist of zero displacement in the radial and tangential directions at the supposed contact line and zero displacement in the radial and axial direction for the front and rear fixing planes of the blade (Figure 7).

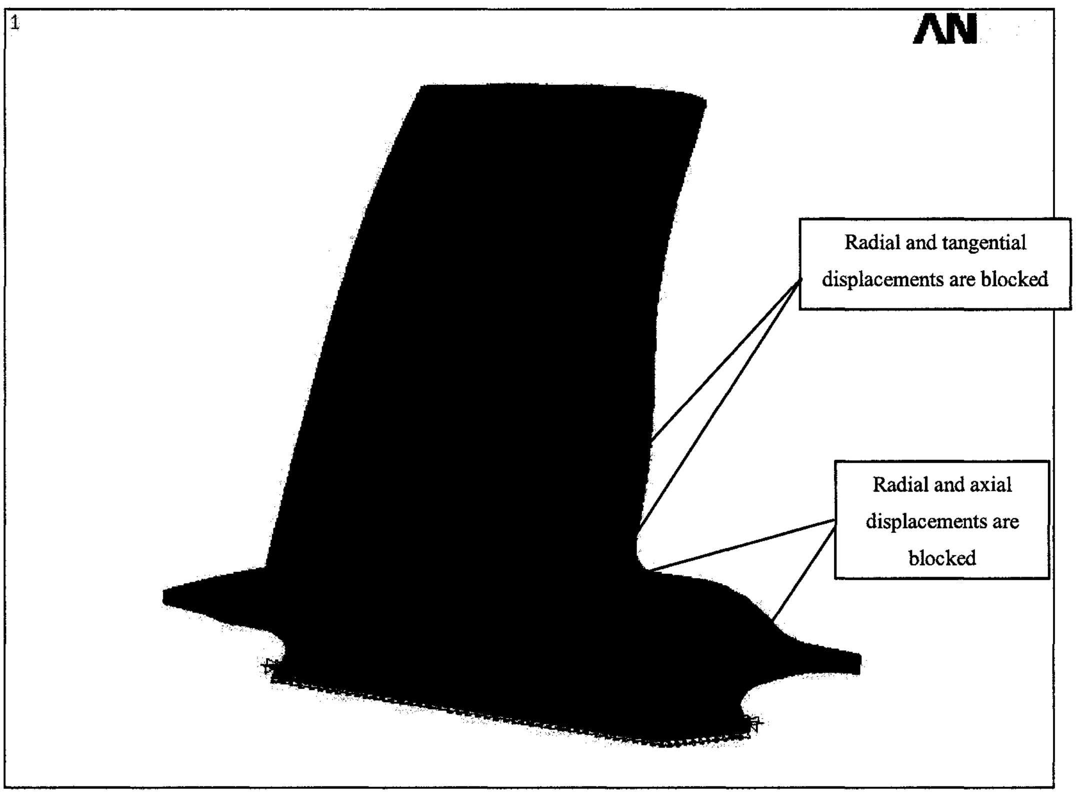

Figure 7 Blade fir-tree line blockage

A static analysis is performed including pre-stress effects such as centrifugal force (rotation) and temperature. The static analysis determines a new mass [M] and rigidity [K] matrices due to the deformation of the blade. Once the static analysis is completed, a modal analysis is performed using the updated matrices. The results from this modal analysis are natural frequencies and mode shapes. Due to the total blockage contact line, there are peak stresses present in the fir-tree area, which do not reflect the reality (Figure 8).

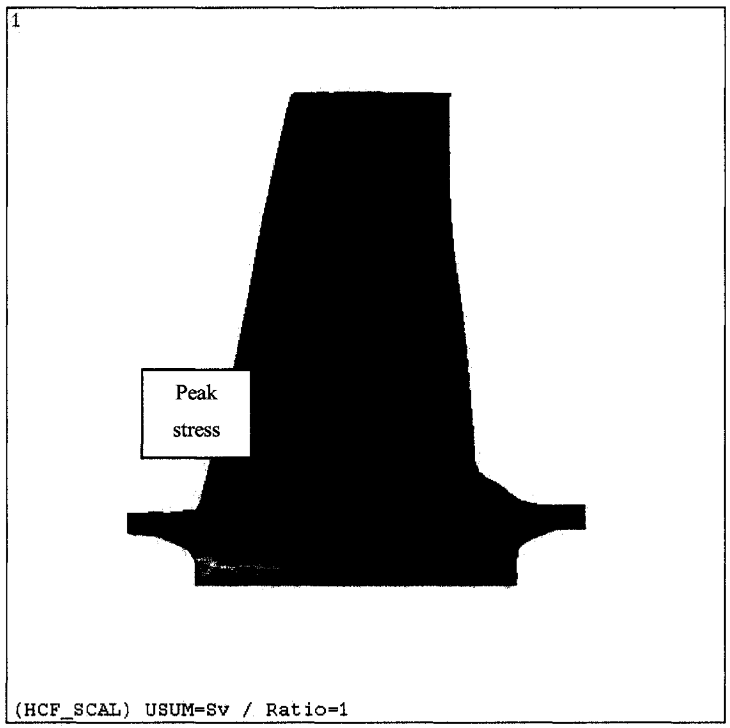

Figure 8 Blade stress with contact lines blocked

The unrealistic stress in the fir-tree area is the main reason for a more realistic modeling using contact elements. The vibratory stress can cause severe damage, which can extend up to the fracture of the blade, at the fir-tree area. Therefore, it is very important to be able to predict with more precision the stresses in that particular region.

4.2 New Analysis

The new analysis will be performed in much the same way, as is the current analysis (section 4.1). A static and modal analysis will be performed sequentially. The difference will be in the boundary conditions settings. In the new analysis, blockage of the blade will not be assumed. Contact elements will be used over the entire fir-tree area, and the static analysis will determine whether displacement occurs. To perform an ANSYS® three-dimensional static contact analysis, two types of elements must be used. The contact element (CONTA174) is used to represent the contact and sliding between the 3-D "target" surfaces and a deformable surface, defined by this element. CONTA174 is an 8-node element intended for general rigid-flexible and flexible-flexible contact analysis. The contact detection points are located either at the nodal points or at the Gauss points. The contact element is constrained against penetration into the target surface at its integration points. However, the target surface can penetrate through into the contact surface. The "target" surface is a geometric entity in space that senses and responds when one or more contact elements move into a target surface. The target element (TARGE170) is used to represent various 3-D target surfaces associated with contact elements. The "contact-target" pair concept has been widely used in finite element simulations.

4.3 Meshing of Contact Elements

The meshing of the components is performed using CATIA®. The section of the disc and the fir-tree region of the blade are meshed using 20-node hexahedral elements. Because of the complex shape of the blade's airfoil, 10-node tetrahedral elements are used for meshing (Figure 9). These analyses typically have approximately 80,000 elements, 100,000 nodes and 300,000 degrees of freedom. The boundary conditions used try to reflect the reality with more accuracy than the current analysis. The sides of the disc portion are fixed in every direction while the blade is fixed in the axial direction using the nodes on the rivet hole to simulate the use of the rivets.

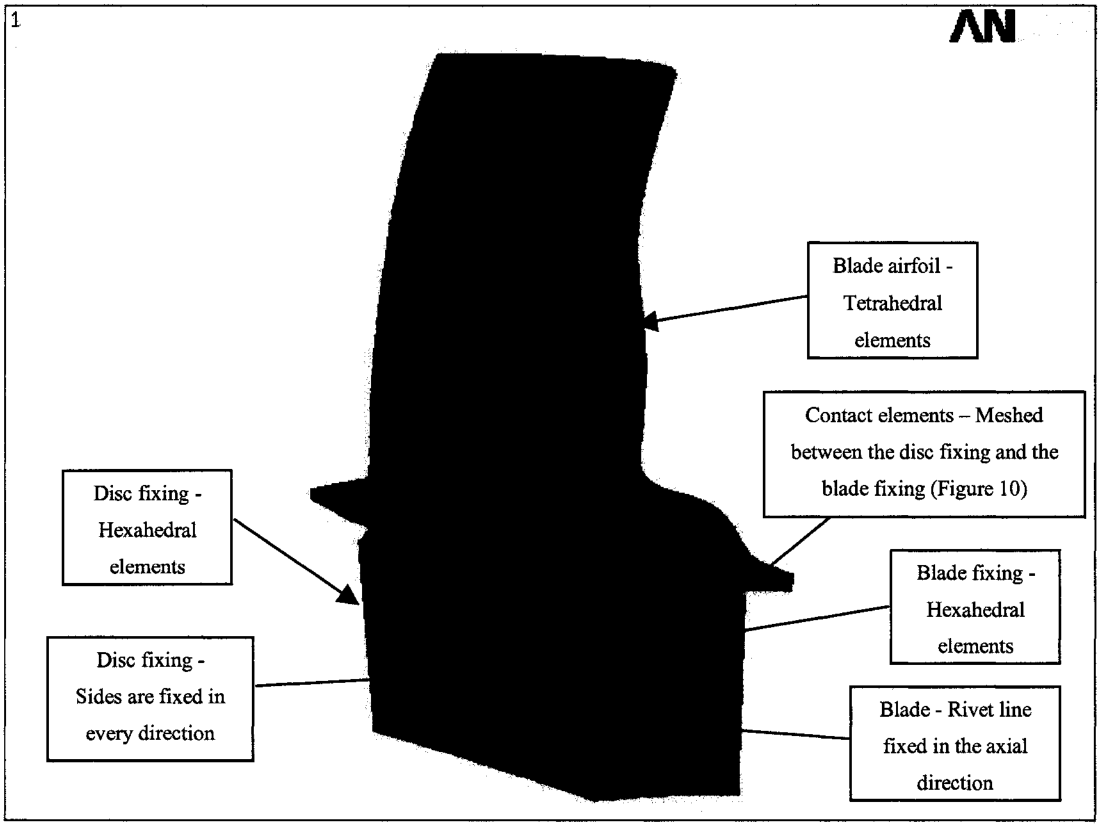

Figure 9 Blade and disc meshing

The meshing of the contact element pairs is performed using ANSYS®. Selections of nodes of the disc and blade fir-tree region in CATIA® are created for meshing purposes in ANSYS®. The target elements (TARGE170) are meshed over the fir-tree area of the disc. There are no assumptions made with respect to the contact areas; and so the whole fir-tree is thus covered with the target elements. The same process is performed for the blade fir-tree region (Figure 10).

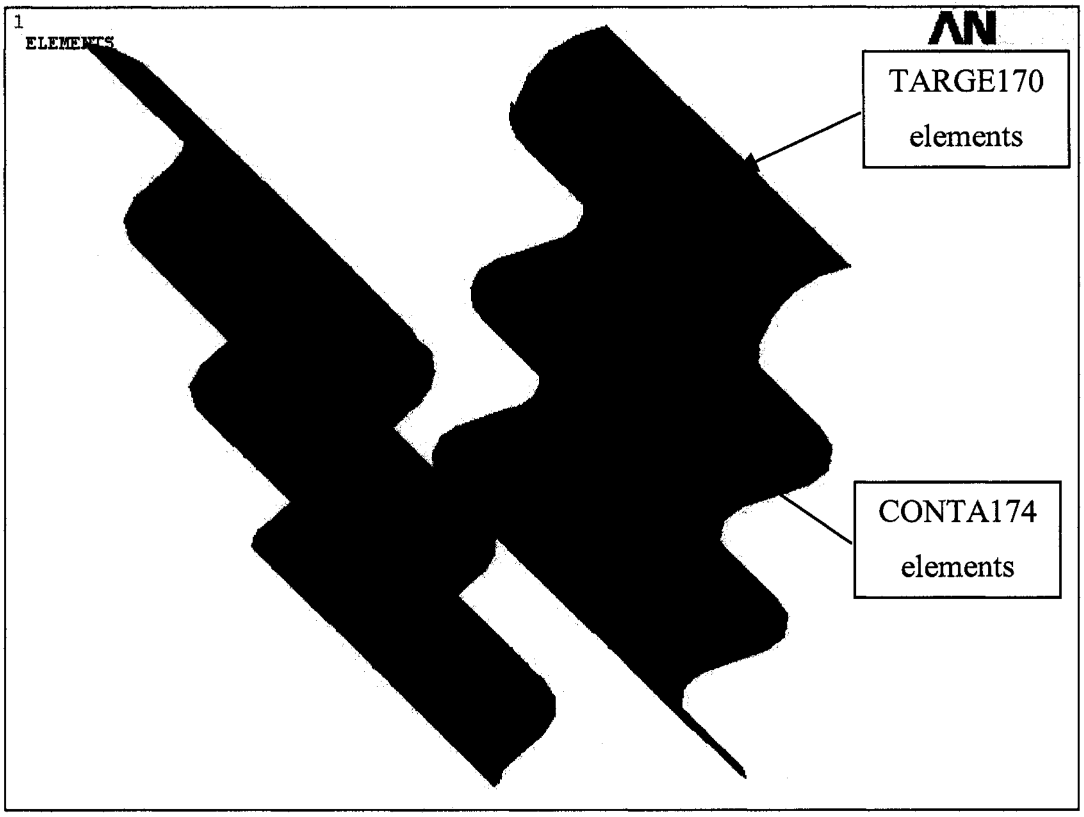

Figure 10 Contact elements mesh

Since the part of the disc has been modeled, it is jugged flexible but no penetration can occur since "target" elements are used. The different contact types of the contact elements during the static and modal analyses are presented in the table below.

Table II

Contact type for the static and modal analyses

Contact Type	Static Analysis	Modal Analysis		
		Initially Touching	Inside Pinball Region	Outside Pinball Region
Bonded	Bonded	Bonded	Bonded	Free
No Separation	No Separation	No Separation	No Separation	Free
Rough	Rough	Bonded	Free	Free
Frictionless	Frictionless	No Separation	Free	Free

Since a friction coefficient value is given to the contact elements, the "rough" contact type occurs during the static and modal analyses for this study. Therefore, after the static analysis is performed, only the elements that are touching to each other will be bonded while the contact elements pairs that are not touching will have no stiffness added.

4.4 Contact Element Input Data

The contact element pair has multiple parameters that have to be defined for the analysis to get a converged solution. The first parameter is the dynamic coefficient of friction (MU), which will have an effect on the limit shear stress and the relative sliding distance. The second and third parameters are the normal contact stiffness factor (FKN) and the penetration tolerance factor (FTOLN) respectively. These factors are related for the convergence purposes. The normal contact stiffness factor determines the penetration rigidity of the component while the tolerance factor determines whether the penetration compatibility is satisfied when using the penalty and Lagrange method. The contact compatibility is satisfied if the penetration is within a tolerance of the FTOLN value multiplied by the depth of the underlying solid. Therefore, if the FKN and the FTOLN factors are too low, the analysis will not converge due to the presence of a higher than allowed level of penetration. The fourth and fifth parameters are required for a smooth transition zone between static and dynamic friction given by the following equation [4.1]:

$$\mu = MU \times \left(1 + \left(FACT - 1\right)\exp(-DC \times V_{rel}\right) \qquad (4.1)$$

μ is the static coefficient of friction, MU is the dynamic coefficient of friction presented previously. The parameters $FACT$ for the ratio between the static and dynamic friction coefficients and DC for the decay coefficient are required. V_{rel} is the slip rate between the blade and the fixing calculated at each time step by ANSYS®. Since these parameters are not known, a convergence study has been performed and will be presented in the following chapter on these five parameters. The contact element pair has many other parameters but they were kept at their default values since they had no effect on frequency values and mode shapes. The results are presented in section 7.2.