Chapter 2 Initial FaBCoM TeSt Finite Element Model

2.1. Purpose

This dissertation is concerned with the development or formulation of a methodology for on-line fan blade damage detection. In order to formulate such a methodology, tests will be done on a laboratory test structure referred to as the FaBCoM TeSt.

In this chapter, a FEM of the FaBCoM TeSt is presented. Smit [45] made use of a FEM of a single fan blade for his dissertation concerning on-line blade damage detection for a single blade. Smit developed the technique using one sensor per blade and concluded that in order to make use of less than one sensor per blade, GMSFs in conjunction with neural networks will need to be used. For this reason, it was decided to develop a more extensive FEM of the FaBCoM TeSt.

This FEM will be used for several purposes namely

- A feasibility study of multiple blade damage detection using GMSFs.
- The identification of GMSFs suitable for experimental on-line damage detection for multiple blades.
- The numerical generation of FRFs for different blade damage levels and cases. Features of the FRFs will be used for training a neural network for damage detection on the FaBCoM TeSt.

Blade damage will be defined as the percentage of blade root crack length to total blade width in this dissertation.

2.2. Fan Blade Condition Monitoring Test Structure

The FaBCoM TeSt was originally named the Experimental Fan Blade Damage Simulator by its designer Smit [45]. The FaBCoM TeSt basically consists of four simple blades with variable pitches, an aluminium and a steel hub as well as a rotor with two radial roller element bearings and a drive pulley. A computer generated isometric assembly drawing of these parts except for the drive pulley is shown in Figure 2-1.

The blades are attached to the aluminium hub which in turn is attached to the rotor by means of the steel hub. These are enclosed by a steel structure and grid for safety reasons as shown in Figure 2-2. The rotor and its bearings are enclosed in a bearing housing that is attached to the steel structure. Drive of the FaBCoM TeSt occurs by means of a fan belt driven by a three phase 1.5 kW electric motor. This is shown in Figure 2-3. The speed of the electric motor is controlled by an AC Tech Variable Speed AC Drive.

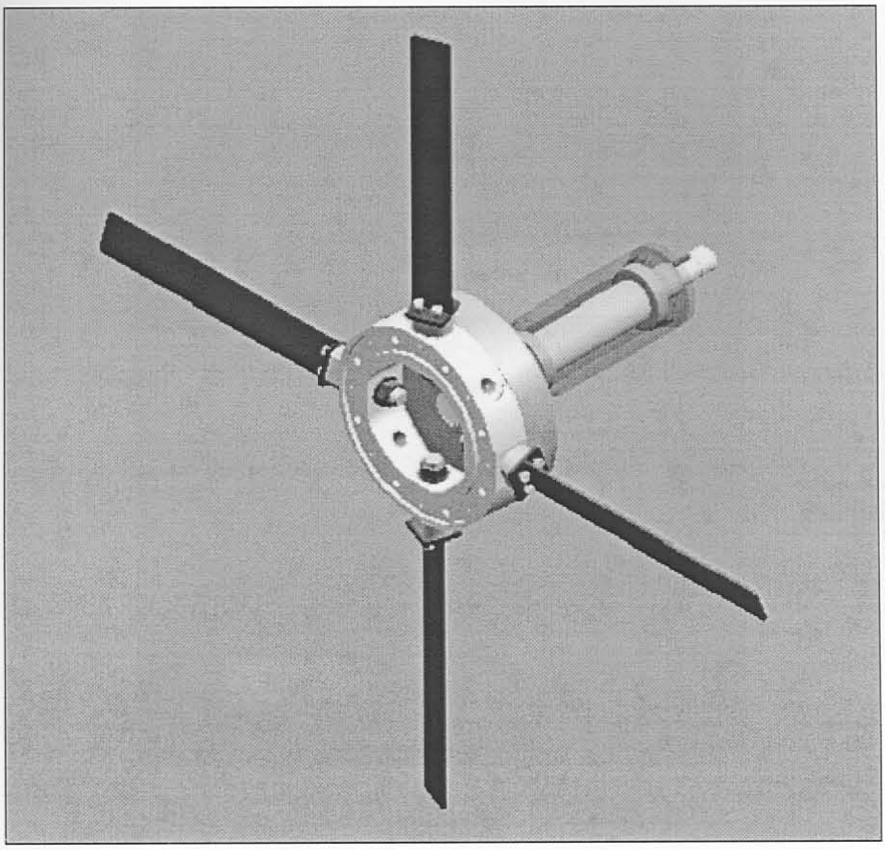


Figure 2-1: FaBCoM TeSt Rotor Assembly Isometric Drawing

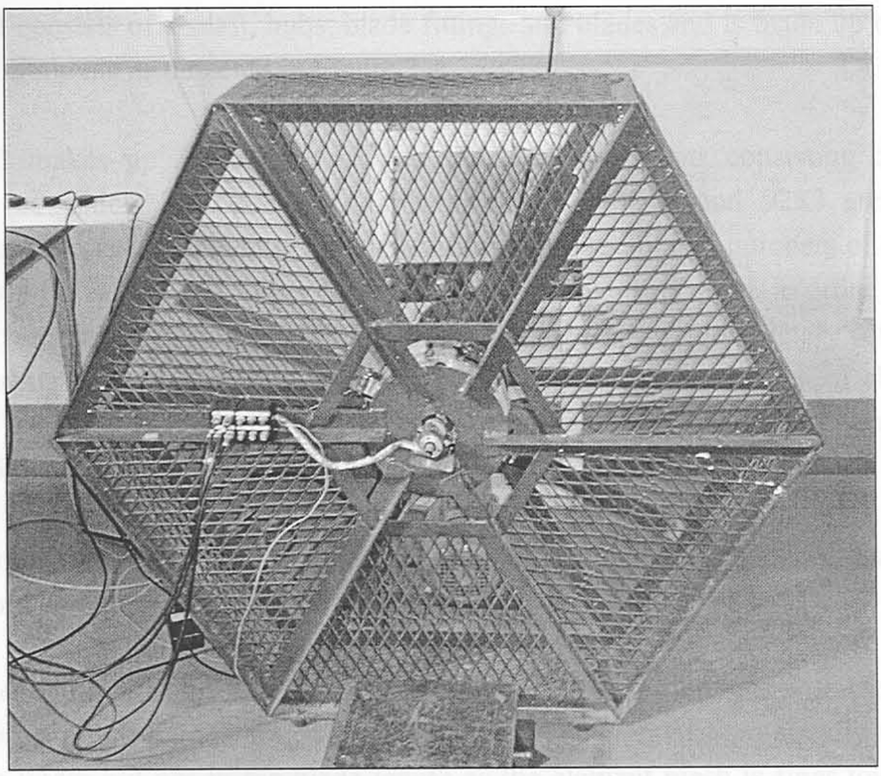


Figure 2-2: FaBCoM TeSt Enclosure

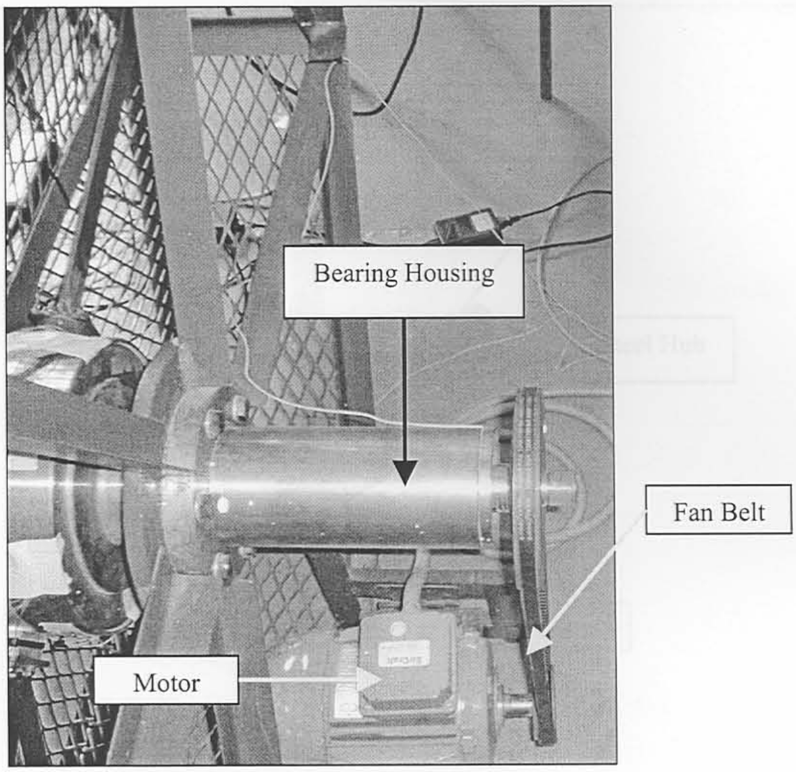


Figure 2-3: FaBCoM TeSt Fan Belt Drive

2.3. *Finite Element Model Description*

The FEM consists of a shaft, hubs, blade fittings and blades and is made up by a total of 13384 elements as depicted in Figure 2-4.

The shaft makes up the biggest part of the model elements consisting of 11464 elements of which 2160 are 8 node hexahedral elements and 9282 are 6 node pentahedral or wedge elements. The reason for using such large numbers of elements is to satisfy the default reliability thresholds as laid down by Patran, in order to obtain accurate results. Still, the reliability thresholds could not be completely satisfied as 113 elements exceed the Edge Angle threshold of 30° by a maximum of about 2.6° and 199 elements exceeds the Face Skew threshold of 30° by a maximum of also about 2.6°.

The aluminium hub consists of 1200 eight node hexahedral elements that satisfy all the Patran element reliability thresholds.

Each blade is made up of 120 four-node quadrilateral shell elements with the element dimensions of each blade being 4 x 30 elements. The elements are equally spaced in the blade width but not in the blade length as the element mesh is finer towards the blade root. Each blade fitting consists of 60 eight-node hexahedral elements. A 10° blade angle is obtained by twisting the hexahedral elements. All these elements satisfy the Patran element reliability thresholds.

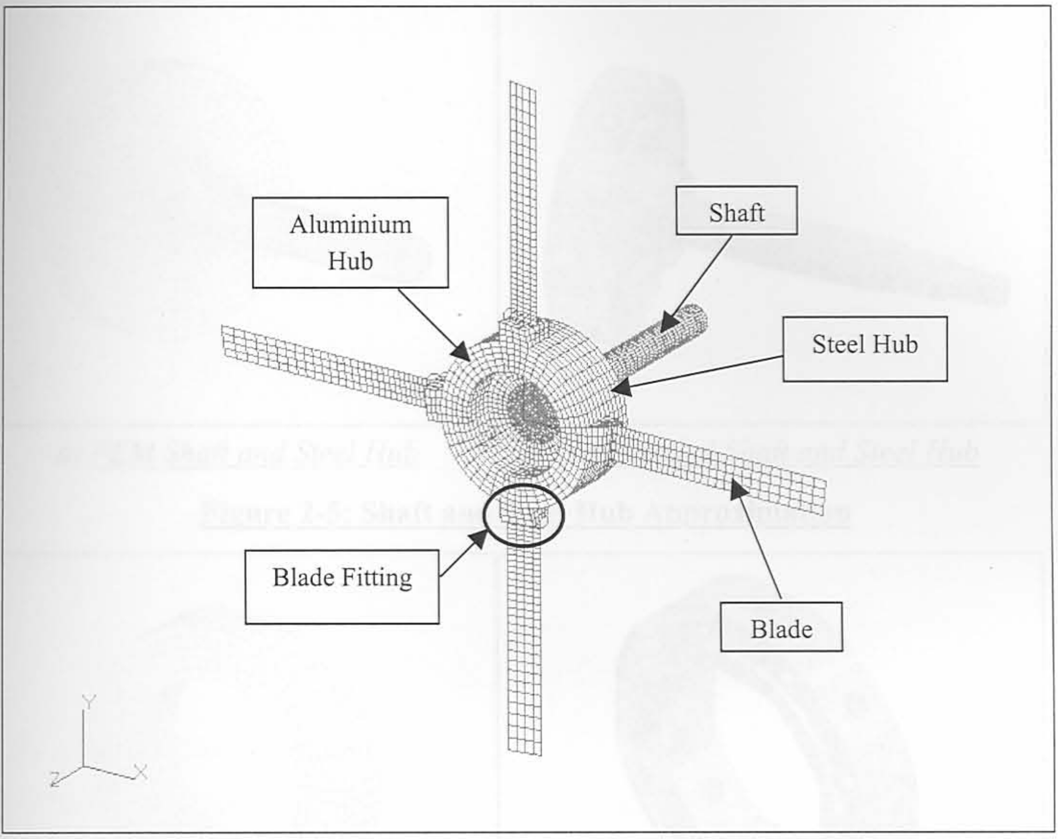


Figure 2-4: FaBCoM TeSt Finite Element Model

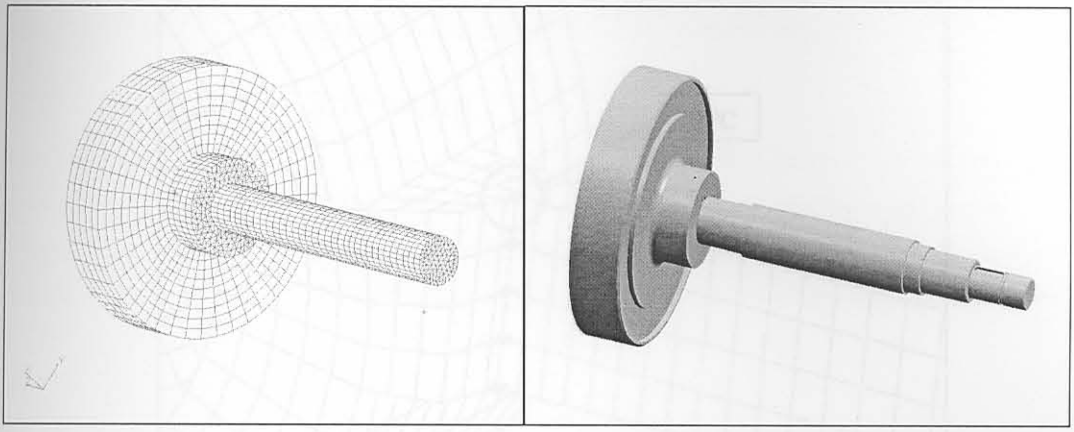
The choice of the specific elements used in the model was made on the basis of model simplicity. It was decided to make use of the simplest forms of the elements, as the element meshes were already more than fine enough to yield accurate results. All the elements used are suitable for static, modal and frequency response analyses ([37]).

2.4. Geometrical Approximations

The FEM is to a large extent a geometrical approximation of the FaBCoM TeSt. The shaft used in the FEM is a simple cylindrical approximation of the actual shaft as shown in Figure 2-5. The shaft diameter used in the FEM is 45 mm, which is the FaBCoM TeSt shaft diameter at the bearing locations.

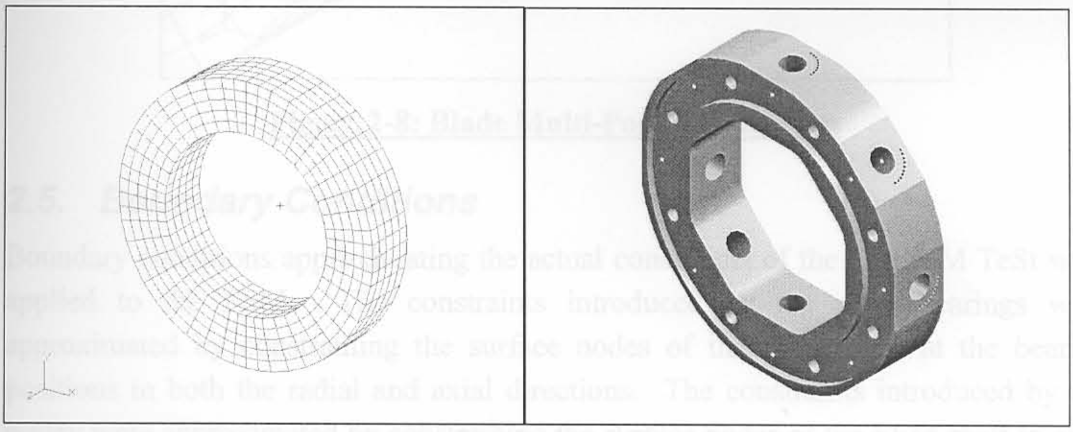
The FEM aluminium hub is a ring approximation of the actual hub as shown in Figure 2-6. A comparison between the actual and approximated blades and blade attachments can be seen in Figure 2-7.

The edge quadrilateral shell elements at the blade roots are connected to the hexagonal elements of the blade fitting by making use of Multi-Point Constraints (MPCs) as shown in Figure 2-8:



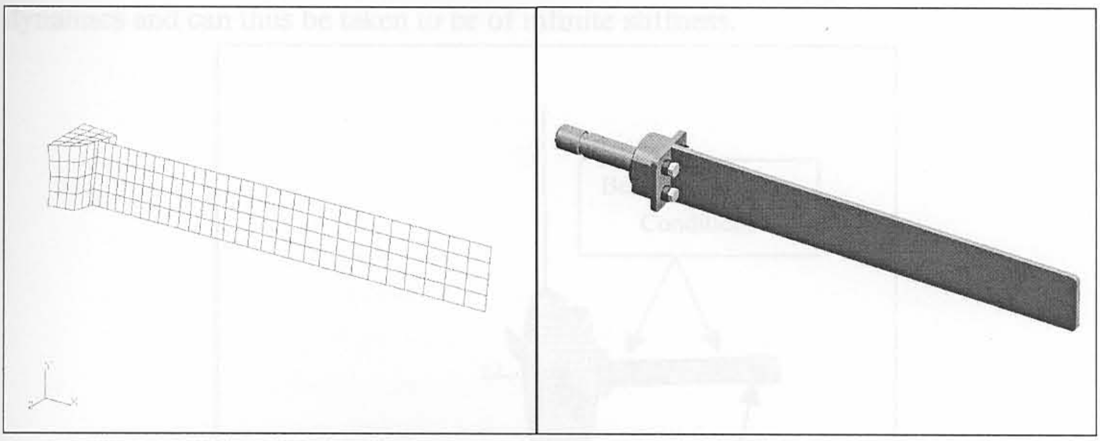
a) FEM Shaft and Steel Hub *b) Detailed Shaft and Steel Hub*

Figure 2-5: Shaft and Steel Hub Approximation



a) FEM Aluminium Hub *b) Detailed Aluminium Hub*

Figure 2-6: Aluminium Hub Approximation



a) FEM Blade and Fitting *b) Detailed Blade and Fitting*

Figure 2-7: Blade and Fitting Approximation

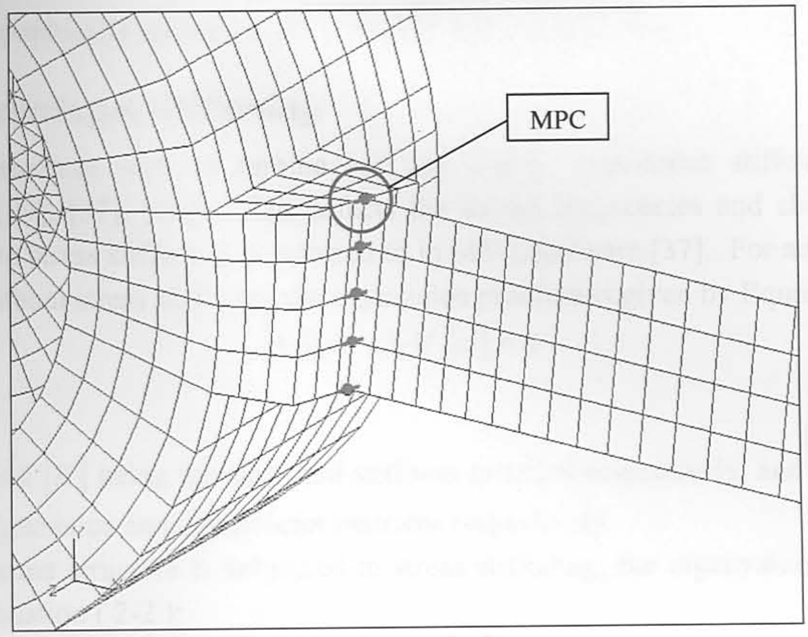


Figure 2-8: Blade Multi-Point Constraints

2.5. Boundary Conditions

Boundary conditions approximating the actual constraints of the FaBCoM TeSt were applied to the FEM. The constraints introduced by the shaft bearings were approximated by constraining the surface nodes of the FEM shaft at the bearing positions in both the radial and axial directions. The constraints introduced by the pulley were approximated by constraining the surface nodes of the FEM shaft in all 6 degrees of freedom. It must be noted that the assumption is made here that the stiffness of the belt used for driving the fan does not contribute to the structure dynamics and can thus be taken to be of infinite stiffness.

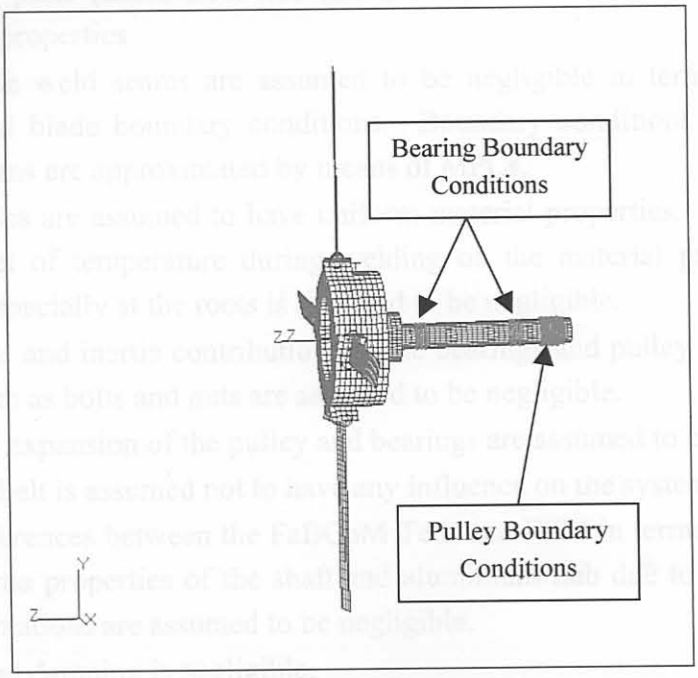


Figure 2-9: Finite Element Model Boundary Conditions

Figure 2-9 graphically represents the FEM boundary conditions.

2.6. Centrifugal Stiffening

Rotating structures such as turbine and fan blades, experience stiffening during operation ([29], [37], [48]). This affects the modal frequencies and shapes of the blades due to stress stiffening as referred to in MSC.Software [37]. For an undamped structure without stress stiffening the eigenvalue problem is given by Equation (2-1):

$$[K]\{\psi\}-\lambda[M]\{\psi\}=0$$

(2-1)

with $[M]$ and $[K]$ being the mass and stiffness matrices respectively, and λ and $\{\psi\}$ being the eigenvalue and eigenvector matrices respectively.

When the same structure is subjected to stress stiffening, the eigenvalue problem is given by Equation (2-2):

$$([K]+[K_g])\{\psi\}-\lambda[M]\{\psi\}=0$$

(2-2)

with $[K_g]$ the geometric stiffness matrix calculated by means of the Theory of Elastic Stability.

Fortunately, it is very easy to implement centrifugal stiffening in Patran.

2.7. Assumptions

This section summarizes the assumptions made during the FEM construction:

- All steel parts (shaft, steel hub and blades) are assumed to have the same material properties
- The blade weld seams are assumed to be negligible in terms of mass and additional blade boundary conditions. Boundary conditions applied by the weld seams are approximated by means of MPCs.
- The blades are assumed to have uniform material properties. In other words, the effect of temperature during welding on the material properties of the blades especially at the roots is assumed to be negligible.
- The mass and inertia contributions of the bearings and pulley as well as other parts such as bolts and nuts are assumed to be negligible.
- Thermal expansion of the pulley and bearings are assumed to be negligible.
- The fan belt is assumed not to have any influence on the system dynamics.
- The differences between the FaBCoM TeSt and FEM in terms of the stiffness and inertia properties of the shaft and aluminium hub due to the geometrical approximations are assumed to be negligible.
- Structural damping is negligible.