Chapter 3

Methodology of Present Study

3.1 Introduction - The Need of a New Approach

In recent years, finite element method has become the preferred tool in studying disc brake squeal by the brake research community. This is owing to the fact that the method offers much faster and more cost efficient solutions than experimental methods and it can predict squeal noise performance at early stage of design development. It also can capture more realistic representation of disc brake, including nonlinearities and flexibilities of the disc brake components. The vast advantages stated above provide a promising future direction of the finite element method against the other methods. However, a lot of work still needs to be done in order for the method to be more reliable and robust in predicting squeal generation.

With the refinements on the methodology and analysis of the disc brake squeal are progressively reported in the literature review, it is thought that refinement in the disc brake model should also be made in parallel. This can be seen from the previous chapter that previous works on the complex eigenvalue analysis required using a number of linear spring elements at the friction interface in order for the stiffness matrix to become asymmetric (Liles, 1989; Ripin, 1995; Lee et al., 1998 and many others). The introduction of a number of spring elements at the disc/pads interface is necessary to generate friction coupling terms (asymmetric stiffness matrix) that lead to the complex eigenvalues i.e. unstable behaviour in which the positive real parts indicate the likelihood of the squeal occurrence. Fortunately, with the contributions of some researchers (Yuan, 1996 and Blaschke et al., 2000) and the initiative of a finite element software company (ABAQUS, Inc., 2003), linear spring elements are no longer required as friction coupling terms can directly implemented into the stiffness matrix. As a result the effect of non-uniform contact pressure and residual stresses can be incorporated in the complex eigenvalue analysis (ABAQUS, Inc., 2003). Another advantage of the current approach is that the surfaces in contact do not need to have matching meshes and essentially it can reduce data-preparation
time. However, the former approach required nodes on the two contacting surfaces to coincide and similar meshes.

Some previous studies assumed full contact at the pads and disc interfaces (Liles, 1989, and Ouyang et al, 2000). However, previous works on the brake contact pressure analysis (Samie and Sheridan, 1990; Tirovic and Day, 1991; Ghesquiere and Castel, 1992; Ripin, 1995; Lee et al, 1998; Hohmann et al, 1999; Tamari et al, 2000 and Ioannidis et al, 2003) has shown that the contact pressure distributions at the disc/pads interfaces are not uniform and there exists partial contact over the disc surfaces. Traditionally, contact analysis at the disc/pads interface was simulated using either linear spring elements (Nack, 2000) or non-linear node-to-node gap elements (Samie and Sheridan, 1990; Tirovic and Day, 1991; Ripin, 1995; Lee et al, 1998, and Tamari et al, 2000). Recent contact analyses (Hohmann et al, 1999; Ouyang et al, 2003a and Ioannidis et al, 2003) no longer use such elements, where a surface based element can provide more realistic and accurate representation of contact pressure distribution (ABAQUS, Inc., 2003).

When developing a finite element model, it is important to validate it in order for the model to correctly represent the actual structure in terms of the geometry and its material properties. A validated model should be able to predict squeal sufficiently accurate. Liles (1989) was the first researcher who conducted and expounded the complex eigenvalue analysis with a large finite element model and used modal analysis to compare natural frequencies and its mode shapes for each of disc brake components. This component validation later became a standard practice (Ripin, 1995; Lee et al, 1998; Guan and Jiang, 1998; Liu and Pfeifer, 2000; Kung et al, 2000 and Ioannidis et al, 2003). Even though precise representations of brake components are possible, results obtained by the complex eigenvalue analysis did not always correspond to the experimental squeal test results. Realising this shortcoming Richmond et al (1996), Dom et al (2003), Ouyang et al (2003a) and Goto et al (2004) used a systematic procedure to correlate and update the FE model at both the component and assembly levels. Tuning process was performed to reduce relative errors in natural frequencies between predicted and experimental results. Material properties and spring stiffness were adjusted in the tuning process.
In the past, simulating disc brake squeal using the complex eigenvalues together with the finite element method was time consuming (Mahajan et al., 1999) compared to the normal mode analysis. In order to speed-up the simulation process ABAQUS, Inc. (2003) creates a new complex eigensolver, which requires four main steps to calculate complex frequencies, whilst Lou et al. (2004) used the adaptive block Lanczos method (ABLE) and Chung et al. (2003) proposed a modal domain analysis.

As progressive refinements on the methodology and analysis stated above are made, it is thought that more realistic finite element model should be developed or, in other words, refinement on the disc brake model should be made. It is already known that contact geometry between the disc and friction material interface has a significant contribution towards squeal generation (Tarter, 1983; Ripin, 1995; Ibrahim et al., 2000; Eriksson et al., 1999 and Sherif, 2004). They believed that squeal could be generated at particular conditions of pads topography. This is true, in fact that the material properties of friction material are much lower than the disc and as a result the friction material is more prone to wear. Furthermore, friction material has much irregular/corrugated surface than the disc. From the literature review, it was found that none of the finite element models considered friction material surface topography. All the models assumed that the friction material had smooth and flat interface whereas in reality it was rough.

As discussed before, most of the FE models are either validated at only component level or combination of the components and assembly levels. In the literature, it was shown that the complex eigenvalue analysis was the most preferred method and most adopted by the industry to study their squeal noise issues. This method is largely dependent on the results of contact analysis, which can determine instability in the disc brake assembly. Determination of dynamic contact pressure through experimental methods remains impossible. However there are methods to obtain static contact pressure, when the disc is stationary. It is believed and will be shown later in this thesis that static contact pressure distribution and its magnitude can be used as a validation tool where correlation between calculated and measured results can be made. Thus, this validation level can enhance one’s confidence on the developed model as well as may provide better prediction in squeal occurrence.
In this research, efforts are focused on the development of a refined FE model, in which brake pad interface is modelled from information of the real surface profile. Measurements on the real friction material surfaces are made using a particular tool described in the following section. Contact tests are also conducted to obtain static contact pressure distribution and the results are used for confirmation of FE results. The effect of wear on the generation of squeal is also simulated to confirm observations found in the experiments by previous researchers. The fugitive nature of squeal is also investigated in the context of friction material surface topography. The complex eigenvalue analysis is extensively used to predict instability in the disc brake. Structural modifications including material modifications are simulated in order to obtain a better disc brake system in terms of squeal noise performance. During the investigation, ABAQUS v6.4 software package is used as the main analysis tool for achieving the objectives of research. The overall simulation scheme is illustrated in figure 3.0.
3.2 Development and Validation Process of an FE Model

This section will describe in detail development and validation process of a disc brake model throughout this research. The first validation stage is using modal analysis, in which natural frequencies and its mode shapes will be compared to the experimental results. It takes two levels of validation, namely, components and assembly. The second validation stage uses contact analysis in which static contact pressure distributions at piston and finger pads, between simulated and measured are compared. There are a number of considerations that should be taken into account in order to develop disc brake components and assembly, and simulate contact analysis. These considerations are given in the subsequent sections.

3.2.1 Construction of a Disc Brake Model

Two commercial software packages are employed in order to generate disc brake model and to simulate the model for particular results. A software package called MSC PATRAN r2001 is utilised to generate elements and nodes of the disc brake. The advantages of using this software are easily changing one element type to another e.g., a linear 8-nodes solid element (C3D8) to a quadratic 20-nodes solid element (C3D20), specifying contact surfaces between the disc and pads, connecting spring elements for components interaction, and simplifying geometry modifications. The software is also able to generate an input file that is compatible with ABAQUS v6.4, which then will be used to perform subsequent analyses such as modal analysis, contact analysis and the complex eigenvalue analysis.

A detailed 3-dimensional finite element (FE) model of a Mercedes solid disc brake assembly is developed. Figure 3.1(a) and 3.1(b) show a real disc brake of floating calliper design and its FE model respectively. The FE model consists of a disc, a piston, a calliper, a carrier, piston and finger pads, two bolts and two guide pins. A rubber seal (attached to the piston) and two rubber washers (attached to the guide pins) are not included in the FE model. Damping shims are also not present since they have also been removed in the squeal experiments. The FE model uses up to 8350 solid elements and approximately 37,100 degrees of freedom (DOFs). This figure excludes the spring elements that have been used to connect disc brake components.
The disc, brake pad, piston, guide pins and bolts are developed using combination of 8-node (C3D8) and 6-node (C3D6) linear solid elements while the other components are developed using combination of 8-node (C3D8), 6-node (C3D6) and 4-node (C3D4) linear solid elements. Element types in the brackets show the notations in ABAQUS nomenclature. Details for each of the components are given in table 3.0.

Since the contact between the disc and friction material surface is crucial, realistic representation of those interfaces should be made. Friction material has a rougher surface and is soft in properties than the disc, which has quite a smooth and flat surface, and is less prone to wear. Therefore in this work, actual surfaces at macroscopic scale of piston and finger pads are considered and measured. A Mitutoyo linear gauge LG-1030E and digital scale indicator are used to measure and provide reading of the surface respectively as shown in figure 3.2. The linear gauge is able to measure surface height distribution ranging from of 0.01mm up to 12 mm.
Table 3.0  Description of disc brake components

<table>
<thead>
<tr>
<th>COMPONENTS</th>
<th>TYPES OF ELEMENT</th>
<th>NO. OF ELEMENTS</th>
<th>NO. OF NODES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Disc</td>
<td>C3D8 C3D6</td>
<td>3090</td>
<td>4791</td>
</tr>
<tr>
<td>Calliper</td>
<td>C3D8 C3D6 C3D4</td>
<td>1418</td>
<td>2242</td>
</tr>
<tr>
<td>Carrier</td>
<td>C3D8 C3D6 C3D4</td>
<td>862</td>
<td>1431</td>
</tr>
<tr>
<td>Piston</td>
<td>C3D8 C3D6</td>
<td>416</td>
<td>744</td>
</tr>
<tr>
<td>Back plate</td>
<td>C3D8 C3D6</td>
<td>2094</td>
<td>2716</td>
</tr>
<tr>
<td>Friction Material</td>
<td>C3D8 C3D6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Guide pin</td>
<td>C3D8 C3D6</td>
<td>388</td>
<td>336</td>
</tr>
<tr>
<td>Bolt</td>
<td>C3D8 C3D6</td>
<td>80</td>
<td>110</td>
</tr>
</tbody>
</table>
Prior to the measurement, the back plate must be flat and level. This can be confirmed by taking four measurement points at both pad abutments and the indicator should show similar height position. Node mapping, as shown in figure 3.2, is required so that surface measurement can be made at particular positions, which are generated from the FE model. By doing this, information that is obtained in the measurement can be used to adjust the coordinates of the piston and finger pad nodes in the brake pad interface model. There are about 227 nodes at the piston pad interface and 229 nodes at the finger pad interface. Since measurement is taken manually, it takes about two and half hours to complete this for a single pad. In this work, three pairs of brake pads (6 pieces) are measured, in which one pair of them are worn while the rest are brand new. The same manufacturer produces all the brake pads. Thus, it is assumed that the global geometry and material properties of the brake pad are the same.

Upon completion of the modelling, all the disc brake components must be brought together to form an assembly model. Contact interaction between disc brake components is represented by linear spring elements (SPRING 2 in the ABAQUS nomenclature) except for the disc/pads interface where surface-to-surface contact elements are employed. This selection is due to the fact that contact pressure distributions at the disc/pad interface are more significant than the other component
contact interfaces. This type of spring element has three-degrees of freedom in the translational direction and the relative displacement across the spring element is the difference of the $i$th component at the spring’s first node and the $j$th component of the spring’s second node:

$$\Delta u = u_i - u_j$$  \hspace{1cm} (3.0)

where $i$ and $j$ are degrees of freedom in the translational direction. Such a spring element allows the users specifying different spring stiffness for different directions.

Figure 3.3 shows a schematic diagram of contact interaction that has been used in the disc brake assembly model. A rigid boundary condition is imposed at the boltholes of the disc and of the carrier bracket, where all six degrees of freedom are rigidly constrained.

![Schematic diagram of contact interaction in a disc brake assembly](image)

**Figure 3.3**  Schematic diagram of contact interaction in a disc brake assembly

### 3.2.2 Modal Analysis

The experimental study of structural vibration has made significant contributions for better understanding in vibration phenomenon and for providing countermeasures in controlling such phenomenon in practice. Typically, experimental observations are always to reach two-fold objectives (Ewins, 1984):

- Determining the nature and vibration response levels
- Verifying theoretical models and predictions
The first measurement objective is referred to as a test where vibration forces or responses are measured during structure normal service environmental operation while the second is a test where the structure or component is vibrated with a known excitation. The second test is much more closely carried out under control conditions and this type of test is nowadays known as modal testing or experimental modal analysis (EMA). There are two different methods of comparison available to verify a theoretical model over EMA. They are a comparison in terms of response properties and modal properties. Although response properties of a tested structure can directly be produced in EMA, it is less convenient for some finite element software packages when it comes to generate frequency response function (FRF) plots. Furthermore, comparisons of modal properties are perhaps most common and convenient in the current practice where natural frequencies and modes shapes (either graphical or numerical) are used to obtain correlation between predicted and EMA results.

In this research, experimental modal analysis that was conducted by James (2003) was utilised to verify the developed FE model. Natural frequencies and graphical mode shapes obtained in the experiments were used to compare with the finite element results. Comparisons are made at two levels, namely, disc brake components model and an assembly model. Firstly, FE modal analysis is performed at components level where the dynamic behaviour of disc brake components in the free-free boundary condition is captured. The second stage is to perform FE modal analysis on the disc brake assembly where the disc and the carrier are mounted to the knuckle. A certain level of brake-line pressure is applied to the stationary disc brake. During the analysis a tuning process (also known as model updating) is required in order to reduce relative errors between the predicted and experimental results. Normally, material properties, such as density and spring stiffness, are tuned or adjusted for disc brake components and assembly models respectively in order to bring closer predicted natural frequencies to the experimental data.

In the FE modal analysis, eigenvalue problem for natural modes of a finite element model in typical matrix notation is given by

\[
(\lambda^2 M + \lambda C + K_s)z = 0
\]  

(3.1)
where $M$ is the mass matrix, which is symmetric and positive definite, $C$ is the damping matrix and $K_s$ is the symmetric stiffness matrix while $\lambda$ and $z$ are the complex eigenvalue and eigenvector - the mode of vibration respectively. When material damping is neglected, $\lambda = i\omega$ in this analysis and the eigenvalue problem is now reduced to:

$$(-\omega^2M + K)z = 0 \quad (3.2)$$

ABAQUS/Standard offers two methods for extracting natural frequency, namely, the Lanczos and the subspace iteration methods. The Lanczos method is generally faster when a large number of eigenmodes are required for a system with many degrees of freedom whilst the subspace iteration method is typically faster when only a few (less than 20) eigenmodes are needed. Since the disc brake model contains more than 37,000 degrees of freedom the Lanczos method is chosen in this analysis.

### 3.2.3 Contact Analysis

After validating the FE model both at components and assembly levels, focus is now concentrated on determination of the realistic contact pressure distribution in the contact analysis. Firstly, contact tests are conducted using a special film that can provide static contact pressure distribution. The tested films are then analysed using an interpretive pressure system in order to obtain both contact pressure magnitude and distribution. This information is used to confirm FE predictions. As stated before, contact pressure is becoming an essential parameter in predicting squeal behaviour especially when either the complex eigenvalue analysis or the transient analysis is employed. Validation at this level is thought to be necessary towards better prediction of dynamic contact pressure distribution and subsequently squeal occurrence.

#### 3.2.3.1 Contact tests

The contact pressure tests are conducted using an in-house disc brake dynamometer. In order to measure static contact pressure distribution, a suitable type of sensor film should be chosen for a particular range of local contact pressure. The film needs to be cut to the shape of the brake pad for it to be well positioned at the pad/disc.
interface. Brake-line pressure at certain levels is then applied to the disc brake for 30 seconds and then removed.

It is not quite sufficient to describe contact pressure distribution by showing only the stress marks on the tested films. The contact pressure distributions should be measured qualitatively and quantitatively for comprehensive understanding. In doing so, a system called Topaq® Pressure Analysis system is used. The Topaq® system is a system that analyses pressure distribution and magnitude from pressure indicating film. The system consists of calibrated densitimetric scanner and Windows software. The system is used to image and interpret the stress marks on the tested film. The system, offered by Sensor Products LLC as a product for purchase or as a service, is accurate to within 4% which is very accurate in the field of tactile pressure measurement. The analyzed image of the film is given in Chapter 4. In the contact tests, all three pairs of the brake pads are used to obtain static contact pressure distribution. Comparison will be made over the predicted results obtained in the FE contact analysis.

3.2.3.2 FE contact analysis

The main components that require generating contact pressure distribution are the disc and the pads. Therefore, a careful consideration should be made on the selection of element type that represents a contact interface. ABAQUS offers three approaches to represent contact interface. They are gap contact elements, surface-to-surface contact interaction and surface-to-node contact interaction. Contact pressure and contact stresses cannot be calculated accurately if surface-to-node interaction is chosen while gap contact elements can only provide contact forces, not stresses. This is not desirable because in order to perform stability analysis using the complex eigenvalue analysis, contact stresses are required for stiffness matrix to be asymmetric. Therefore, surface-to-surface contact interaction is the most suitable method and is chosen in this analysis.

For surface-based contact, a master and a slave surface are required to form a contact pair. In order to conduct contact analysis between two deformable components with large rotation of the disc, the master-slave surface approach is suitable. Since the disc is much stiffer and has coarser mesh it is chosen as the master surface and on the
other hand the pads provide the slave surface. Convergence problem could occur if there are large over-closures during the first step, i.e., during the application of the brake line pressure. Therefore it is necessary to make sure any nodes at the slave surface do not initially penetrate the master surface.

There are three types of contact scheme available in ABAQUS namely, small, finite and infinitesimal sliding. By default, ABAQUS treats finite sliding by which contact surfaces may allow for arbitrary separation, sliding and rotation. Using finite sliding, the slave nodes may come in contact anywhere along the master surface and the load transfers are updated throughout the analysis. Whilst for small sliding the contact formulation assumes that the contact surfaces may undergo arbitrarily large rotations but that a slave node will interact with the same local area of the master surface throughout the analysis. Therefore the slave nodes are not monitored that in contact along the entire master surface. With the finite and small sliding consider geometric nonlinearities, infinitesimal sliding ignores this effect and assume both relative motions and the absolute motions of the contacting bodies are small. Thus, infinitesimal sliding is not suitable for disc brake analysis. In this work, it is assumed that the pads will interact with the same profile of the rotating disc surface. Thus, small sliding scheme is chosen. Convergence could also be easily obtained, compared with the finite and infinitesimal sliding formulation. Furthermore, small sliding scheme provides significant computational savings over the finite sliding model.

ABAQUS defines the contact pressure between the surfaces at a point, \( p \), as a function of the over-closure, \( h \), of the surfaces. For this work a hard contact model is considered where the pad and disc surfaces will separate (or contact constraint is removed) when the contact pressure between them becomes zero or negative and on the other hand, the pad and disc surfaces will interact (or contact constraint is applied) when the contact pressure between them is larger than zero. Two regimes for \( p = f(h) \) are given in the formulations below (ABAQUS, Inc., 2003):

\[
\begin{align*}
    p &= 0 & \text{for } h < 0 & \text{(open)} \\
    h &= 0 & \text{for } p > 0 & \text{(closed)}
\end{align*}
\]  

(3.3)
When surfaces are in contact they usually generate shear (friction) and normal forces across the sliding interface. A relationship between these two force components is described in terms of friction between the contacting bodies. Typically, when deriving friction in a theoretical context, the critical value of tangential force is defined as:

\[ F_{\text{crit}} = \mu N \]  

(3.4)

where \( F_{\text{crit}} \) is the critical shear force, \( \mu \) is the friction coefficient and \( N \) is the normal force. Due to the discretisation process used by the finite element method, the critical value is not defined in terms of a critical load \( (F_{\text{crit}}) \), but as a critical shear stress \( (\tau_{\text{crit}}) \) that is a function of pressure \( (p) \) as given below:

\[ \tau_{\text{crit}} = \mu p \]  

(3.5)

The value of shear stress that compares with the critical value, defined above, is the magnitude of the resultant shear stress in the \( x \) and \( y \)-directions:

\[ \tau_{eq} = \sqrt{\tau_x^2 + \tau_y^2} \]  

(3.6)

If the value of the equivalent shear stress is greater than value of the critical shear stress sliding contact will be initiated and the restoring shear stress will be equivalent to \( \tau_{\text{crit}} \). In the case of sticking condition, the shear stress will balance that applies to the contact interface.

ABAQUS provides various friction models to govern relative tangential motions of contact surfaces. A basic Coulomb friction model is used where, by default, friction coefficient can be defined as a function of sliding speed, contact pressure and average temperature at the contact point. The users can also specify different friction coefficients i.e., static friction coefficient and kinetic friction coefficient. In this model it is assumed that the friction coefficient decays exponentially from the static value to kinetic value based on the following equation:
\[ \mu = \mu_k + (\mu_s - \mu_k) e^{-d_c \nu} \]  

(3.7)

where \( \mu_k \) is the kinetic friction coefficient, \( \mu_s \) is the static coefficient, \( d_c \) is a decay coefficient and \( \nu \) is the sliding speed. In specifying static and kinetic friction coefficients, ABAQUS allows the users to change the friction coefficient during an analysis. This is adopted in this entire research where static friction coefficient is used during the first step and kinetic friction coefficient in the next steps. ABAQUS also provides an anisotropic friction model that allows the users specify different friction coefficients in the two orthogonal directions on the contact surface. The users are also allowed to develop their own friction model using user-defined subroutine.

There are two stiffness methods for friction constraints available in ABAQUS, namely, a penalty method and the Lagrange multiplier method. The penalty method (default by ABAQUS) permits some relative motion of the surfaces when the surfaces should be sticking whilst the Lagrange method should be used when no slip is allowed in sticking condition. Using the Lagrange method can increase the computational cost of the analysis since it adds more degrees of freedom to the model and quite often increases the number of iterations to obtain a converged solution. In addition, the Lagrange formulation may prevent convergence of a solution. Therefore in this work the penalty method is employed to ease convergence restriction as well as to obtain minimum computational cost.

### 3.3 Stability Analysis

The complex eigenvalue analysis made available in ABAQUS is utilised to determine disc brake assembly stability. The positive real parts of the complex eigenvalues indicate the degree of instability of the disc brake assembly and are thought to indicate the likelihood of squeal occurrence. The essence of this method lies in the asymmetric stiffness matrix that is derived from the contact stiffness and the friction coefficient at the disc/pads interface (Liles, 1989).

The transient analysis is not as mature as the complex eigenvalue analysis in the finite element method. Instability in the disc brake system through this analysis can be found with an initially divergent vibration time response. This time domain
information then can be converted to frequency domain information using the Fast Fourier Transform (FFT) technique. The transient analysis seems to be more practical for small-sized models with the analytical method as it is highly computationally intensive as reported in the open literature. It has been reported that the transient analysis is also capable of predicting squeal quite well with large degrees of freedom model (Travis, 1995; Hu et al, 1999 and Auweraer et al, 2002). However, very often only one analysis method was performed at one time to predict squeal and compared it with the experimental method results. It is therefore of interest to see the correlation between the two methods for a large degrees of freedom model particularly using a commercial software package such as ABAQUS.

Typically, the complex eigenvalue analysis is simulated in the implicit version while the explicit version is used for the transient analysis. Due to limitations in the explicit version, a reduced FE model is used in this investigation. This is to make sure that the element types and its boundary conditions remain the same for both analysis methods. Comparison is made for a perfect surface and an irregular surface of the brake pad.

In order to perform the complex eigenvalue analysis using ABAQUS, four main steps are required (Kung et al, 2003). They are given as follows:

- Nonlinear static analysis for applying brake-line pressure
- Nonlinear static analysis to impose rotational speed on the disc
- Normal mode analysis to extract natural frequency of undamped system
- Complex eigenvalue analysis that incorporates the effect of friction coupling

In this analysis, the complex eigenvalues are solved using the subspace projection method. The eigenvalue problem can be given in the following form:

\[(\lambda^2 M + \lambda C + K)y = 0\]  \(\text{(3.8)}\)

where \(K\) is the unsymmetric (due to friction) stiffness matrix. This unsymmetrical stiffness matrix leads to complex eigenvalues and eigenvectors. In the third step stated above, the symmetric eigenvalue problem is first solved, by dropping damping
matrix \( \mathbf{C} \) and the unsymmetric contributions to the stiffness matrix \( \mathbf{K} \), to find the projection subspace. Therefore the eigenvalue, \( \lambda \) becomes a pure imaginary where \( \lambda = i \omega \), and the eigenvalue problem now is similar to the equation (3.2).

This symmetric eigenvalue problem then is solved using subspace eigensolver (ABAQUS, 2003). The next step is that the original matrices are projected in the subspace of real eigenvectors and given as follows:

\[
\mathbf{M}^* = [\mathbf{z}_1, \mathbf{z}_2, \ldots, \mathbf{z}_n]^T \mathbf{M} [\mathbf{z}_1, \mathbf{z}_2, \ldots, \mathbf{z}_n], \tag{3.9a}
\]

\[
\mathbf{C}^* = [\mathbf{z}_1, \mathbf{z}_2, \ldots, \mathbf{z}_n]^T \mathbf{C} [\mathbf{z}_1, \mathbf{z}_2, \ldots, \mathbf{z}_n], \tag{3.9b}
\]

\[
\mathbf{K}^* = [\mathbf{z}_1, \mathbf{z}_2, \ldots, \mathbf{z}_n]^T \mathbf{K} [\mathbf{z}_1, \mathbf{z}_2, \ldots, \mathbf{z}_n], \tag{3.9c}
\]

Now the eigenvalue problem is expressed in the following form:

\[
(\lambda \mathbf{M}^* + \lambda \mathbf{C}^* + \mathbf{K}^*)\mathbf{y}^* = 0 \tag{3.10}
\]

The reduced complex eigenvalues problem is then solved using the QZ method for a generalized nonsymmetrical eigenvalue problem. The eigenvectors of the original system are recovered by the following:

\[
\mathbf{y}^k = [\mathbf{z}_1, \mathbf{z}_2, \ldots, \mathbf{z}_n] \mathbf{y}^{**} \tag{3.11}
\]

where \( \mathbf{y}^k \) is the approximation of the \( k \)-th eigenvector of the original system.

The linearised shear stress that takes into account friction effect and the inclusion of \( \mu - \nu \) characteristics is expressed in the following form (Bajer et al, 2003):

\[
d\tau_i = \left( \mu + \frac{\partial \mu}{\partial p} p \right) n_i dp + \frac{\partial \mu}{\partial \dot{\gamma}} n_i n_j d\dot{\gamma}_j + \frac{\mu p}{\dot{\gamma}} (\delta_{ij} - n_i n_j) d\dot{\gamma}_j \tag{3.12}
\]

where

- \( \tau_i \) - tangential stress,
- \( \mu = \mu(\dot{\gamma}, p) \) - friction coefficient,
- \( p \) - contact pressure,
\[ n_i = \frac{\gamma_i}{\gamma}, \quad i = 1,2 \] - normalised slip direction (1 is the radial direction and 2 is the circumferential direction)

\[ \tilde{\gamma} = \sqrt{\gamma_1^2 + \gamma_2^2} \] - equivalent slip rate/sliding velocity,

\[ \dot{\gamma} \] - relative slip/sliding velocity.

The first term in equation (3.12) is the friction contribution that yields unsymmetrical stiffness matrix and essential to capture the phenomenon of mode coupling. The second term is contributing to the damping matrix and only exists if friction depends on the relative slip rate or sliding velocity. If there is a negative friction-velocity gradient exists, this contribution may lead or trigger instabilities. The last term in the expression is also contributing to the damping matrix. However, it is caused by the friction forces that stabilise the vibration along the contact surface. This term is proportional to the contact pressure but inversely proportional to the slip velocity. The significance of this term is that it may suppress some of unstable modes.

Comparing to the other commercial finite element software packages, which require user-defined subroutine to include friction coupling, ABAQUS provides automatic friction coupling matrix that allows straightforward brake squeal analysis. This new functionality requires no coincident nodes between the contact surfaces, which can reduce time in developing the FE model.

### 3.4 Consideration of Wear Effect

It is already known that wear can affect contact pressure distribution and squeal occurrence. It has also been hypothesised that due to wear or real surface topography squeal might be triggered or vanished within the same system and seemingly identical operating conditions in different brake dynamometer tests. For instance, Eriksson et al (1999) and Sherif (2004) stated that squeal could appear and disappeared based upon the condition of brake pad surface topography, which was directly related to wear. There have been experimental studies of wear effect on squeal generation. But simulation of wear effect using the finite element method is rather laborious to perform. This can be seen from the simulation works that have been published. Wear simulation has been performed by Podra and Anderson (1999)
and Kim et al (2005) which are not dedicated to the brake system. They used generic Archard wear formulation in order to obtain wear displacement. Using this information, the FE nodes are adjusted to represent material removal. In a recent simulation work of disc brake squeal using ABAQUS v6.5 by Bajer et al (2004), they included wear effect where a simple wear function was used. They defined the wear displacement as:

\[ w = kp \]  \hspace{1cm} (3.13)

where \( w \) is the wear displacement in the normal direction of the brake pad, \( k \) is the constant and \( p \) is the contact pressure. However, the wear displacement is limited to very small amount i.e. 0.005 mm. Similar to the work by Podra and Anderson (1999) and Kim et al (2005) the coordinates of the interface nodes were adjusted in the normal direction in the current project. A user-defined subroutine was necessary to create a new surface profile.

In this research, wear effect is simulated and a similar methodology presented in (Podra and Anderson, 1999 and Kim et al, 2005) is adopted. Rather than using a simple wear function as defined by Bajer et al (2004), a wear rate formula that proposed by Rhee et al (1970) is used which can be simplified to the following form:

\[ h = kpr\Omega t \]  \hspace{1cm} (3.14)

where \( h \) is the wear displacement and \( p \) is the contact pressure, \( k \) is the specific wear rate coefficient, \( r \) is the mean radius of the pad, \( \Omega \) is the disc rotational speed and \( t \) is the disc sliding time. Contact analysis is first simulated to obtain contact pressure. This information is then used to calculate wear displacements, which then are used to adjust FE nodes at the pad interface. These adjustments are made in the ABAQUS input file. This is quite different from the Bajer et al’s simulation methodology. Using this procedure, different pad surface topographies are generated in one analysis in current project while the same surface topography is generated in Bajer et al’s work (2004).
3.5 Squeal Suppression Methods

Up to date, there are a number of techniques and methods that have been used in order to reduce or eliminate “annoying” squeal and to keep warranty claims as low as possible as poor noise performance would reduce profit margin. The most popular methods that have been thought to effectively control or reduce squeal and currently adopted in the industry are using damping shims, chamfered pad at both edges, slotted pad, applying grease at the guide pins and many more. Apart from those, there are methods that actively control the squeal occurrence. They are “smart” pads design (von Wagner et al, 2004) and dither control (Cunefare and Graf, 2002), both of which require rather expensive investment and probably are not feasible for a mass-produced brake system.

There are also methods other than those stated above that have been studied and reported in the open literature. These reduction techniques are listed out in the previous chapter in Section 2.7. In this work, several structural modification ideas including material modification are proposed and simulated in order to determine their effectiveness against squeal occurrence. Having obtained a good modification, it then will be examined under various brake operating conditions and under variation of Young’s modulus of the disc brake components.

3.6 Summary

A new methodology in predicting disc brake squeal is proposed and developed. Previous studies using the finite element method models are only validated at two validation levels, namely, components and assembly models using modal analysis. In this work, the developed FE model is validated at three-validation stages that also include static contact pressure. The predicted results are compared with the contact test results. Three pairs of brake pads that include two brand new pairs of pad and one worn pair of pad are tested. In the contact tests, a special film is used and then optically analysed using a post-process interpretive system of Sensor Product LLC.

In the past all FE models assumed the presence of perfect pad interface. In fact, the real surface topography of the brake pad is irregular and rough. Assuming a perfect surface may lead to unrealistic contact pressure distribution and in turn may produce
large discrepancies in predicting squeal occurrence. It is quite often that the complex eigenvalue analysis is considered as rather a diagnostic tool than a predictive tool. This is due to the fact that this analysis method produces over-prediction and may miss unstable modes in the squeal frequency range. In order to overcome these drawbacks, a real surface topography of the brake pad is measured at macroscopic scale. This information is used to model realistic surface profile of the brake pad. FE nodes at the pad interface are adjusted in the normal direction by changing its coordinates.

Having obtained the validated FE model and considered a real surface topography of the brake pad, complex eigenvalue analysis in the absence of thermal effect and material damping is performed in order to determine instability of the disc brake system. Numerical results are compared with the experimental test data for a wide brake-line pressure range. Considerations in the analysis may include the inclusion of negative damping or constant friction coefficient and friction damping, and lastly a comparison between a perfect surface profile and real surface topography is made.

It has been suggested that wear may affect squeal generation. In order to confirm this claim, wear is included in the FE model, which is calculated from a wear formula explained in Section 3.2. A number of wear simulations (at steady state) are run and updated surface topography is constructed after each of contact analysis iteration. Using these different surface topographies, the complex eigenvalues are computed to see the variation of unstable frequencies.

To complete the investigation in this research, a comparison between the complex eigenvalue analysis and the transient analysis is made. The reduced FE model is used in the comparison due to limitations in the explicit version of ABAQUS as well as to reduction in computational time. In this work, several structural modifications are also simulated in order to obtain a method of squeal suppression. Suggestion of a good modification is made that may help engineers to develop a quieter disc brake system as well as to improve squeal noise performance.