Chapter 3

Materials and methods
CHAPTER 3
MATERIALS AND METHODS

This study was conducted using a three dimensional finite element analysis. The finite element method (FEM) is a numerical method of analysis for stresses and deformations in structures of any given geometry. The structure is discretized into the so called ‘finite elements’ connected through nodes. The type, arrangement and total number of elements affect the accuracy of the results. The FEM has become one of the most successful engineering computational methods and most useful analysis tool since the 1960s.\textsuperscript{98}

Four different types of implant abutment interface designs of commercially available implant systems were initially were studied using FEA. The designs included

1. **TCC** - The tri channel cylinder connection of Nobel Replace. Fig: 3.1
2. **CC2** - The two piece conical connection of Nobel Active. Fig: 3.2
3. **CC1** - The single piece conical connection (without the abutment screw) of Osstem Implant system. Fig: 3.3
4. **PHC** - The parallel 6 point hex connection of Frialit II Implant system. Fig: 3.4

Based on the analysis of the current designs one new novel design (NC) Fig: 3.5 was arrived at and two variations of this model were modelled their response to the same loading conditions were analyzed using FEA. Two models of this design were:

5. **NC1** – A two piece conical connection with an engaging coronal tri channel feature.
6. **NC2** – A two piece conical connection with a non-engaging coronal tri channel anti-rotational feature.
3.1 Construction of the Geometrical Models:

Creating an accurate analytical model of a dental implant, using appropriate engineering software, is essential in producing realistic and reliable solutions. Accurate and efficient modelling can provide insight and understanding of the complicated nature of a dental implant that is surrounded by the jawbone. The success of modelling depends on the accuracy in simulating the geometry and surface structure of the implant, the material characteristics of the implant and jawbone, the loading and support conditions as well as the biomechanical implant-jawbone interface. Following this, a redesign of the implant geometry can be performed to achieve an optimum stress profile in the surrounding jawbone.

The geometrical models of four implant abutment assembly were constructed using reverse engineering technique in PRO-ENGINEER 05(Parametric Technology Corporation) through three-dimensional optical scanning and point cloud data extraction. FIG.3.1, 3.2, 3.3 and 3.4. One novel design was created using CAD drawings FIG 3.5. Two variations of this new model were included in the study NC1 and NC2. The entire outer abutment surface of NC1 interface was considered to be united with the inner implant surface, whereas in the NC2 interface only the conical portion of the outer abutment surface united with the inner implant surface and a gap was incorporated at the tri channel portion of the interface.

Lambrecht JT et al., (2009) in their study indicated that a model generated from a CT scan can provide results closer to a real scenario; and found a difference in the behavior of stresses in work conducted with elliptical models, cobblestones, and CT scan data. Thus, a CT scan was used as a reference to model the geometry of posterior mandibular region. The bone encountered in the posterior mandible, classified as Type 2 bone, was described as a thick layer of compact bone surrounding a core of dense trabecular bone. A bone block model was constructed based on a cross-sectional image of the human mandible in the pre-molar region, 25 mm high, 12 mm wide and 10 mm thick, consisting of a spongy center surrounded by a 2 mm cortical bone. The implant abutment assembly model was then positioned in the cortical and cancellous bone model. Complete or 100% osseointegration at the implant-bone interface was simulated by sharing the nodes of the elements at this interface.
FIG 3.1: Tri Channel Connection TCC
FIG 3.2: Two Piece Conical Connection CC2

Width of the prosthetic aspect of abutment (mm)

Total Length of the abutment assembly (mm)

Height of the abutment (mm)

Gingival Height (mm)

Height of the cone (mm)

Height of the Hex (mm)

CORONAL VIEW OF THE INTERNAL SURFACE OF THE IMPLANT
FIG 3.3: One Piece Conical Connection CC1

CORONAL VIEW OF THE INTERNAL SURFACE OF THE IMPLANT
Width of the prosthetic aspect of abutment (mm)

Total Length of the abutment assembly (mm)

Gingival Height (mm)

Height of the abutment (mm)

Height of the Hex (mm)

Height of the cylinder (mm)

CORONAL VIEW OF THE INTERNAL SURFACE OF THE IMPLANT

FIG 3.4: Parallel Hex Connection PHC
FIG 3.5: New tri channel Conical Connection NC
3.2 Meshing of the Finite element model:

Numerical representation of the geometry were created by dividing the geometry into finite number of elements and these elements were connected together with nodes leading to conversion of the geometric model into the finite element model. The three dimensional finite element models corresponding to the geometric model were meshed using HYPERMESH 11 (Altair Hyper works). The basic principle of the FEA is to divide a problem domain into non overlapping finite number of small elements. It is achieved by replacing the continuum by a set of key points called as Nodes, which when properly connected form the elements. This collection of nodes and elements forms the finite element mesh. The mesh generation can be carried out either manually or automatically. The number of elements used in a problem depends mainly on the element type and accuracy desired. As a general rule, the larger the number of nodes and elements, the more accurate is the finite element solution. The total number of nodes and elements for each of the Implant abutment assembly is indicated in the Table 3.1

All the components were meshed with SOLID 92 elements. SOLID 92 is a 2nd order Tetra Element which has 10 Nodes (Fig.3.6) Solid 92 have a quadratic displacement behavior and is well suited to model irregular surfaces. The element is defined by ten nodes having 3DOF (degree of freedom) at each node, 3 translations in nodal X, Y, and Z directions. Solid 92 Element mainly have plasticity, creep, swelling, stress stiffening, large deflection and large strain capabilities.

Fig. 3.6.2nd order tetra element with 10 nodes having a quadratic displacement behavior.
This meshed model was then imported into ANSYS 13 software (Ansysinc., USA) to perform the numerical simulation.

### 3.3 Boundary conditions and constraints:

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>YOUNGS MODULUS (MPA)</th>
<th>POISONS RATIO</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cancellous bone$^{104-106}$</td>
<td>1100</td>
<td>0.30</td>
</tr>
<tr>
<td>Cortical bone$^{104-106}$</td>
<td>13700</td>
<td>0.30</td>
</tr>
<tr>
<td>Titanium (implant)$^{107,84}$</td>
<td>110,000</td>
<td>0.33</td>
</tr>
<tr>
<td>Titanium alloy (abutment and screw)$^{107,84}$</td>
<td>110,000</td>
<td>0.33</td>
</tr>
<tr>
<td>Cobalt chromium (metal coping)$^{108}$</td>
<td>87900</td>
<td>0.30</td>
</tr>
<tr>
<td>Porcelain$^{109-111}$</td>
<td>70,000</td>
<td>0.19</td>
</tr>
</tbody>
</table>

The different structures used in the finite element model were assigned their respective material properties. Young’s modulus and Poisson’s ratio are the two properties that were considered for the study.
In this study, it was assumed that the implant, abutment, screws and the porcelain fused to the metal crowns had homogenous, linear elastic, and isotropic elastic properties. But cortical and cancellous bones were treated as anisotropic. Material properties for bone implant abutment complex and crown (as summarized in the table 3.2) were collected from reliable resources and published data. The implant was pure titanium and other components were titanium alloys, and the PFM crown contained cobalt chromium metal (coping) with porcelain having homogenous and isotropic elastic properties.

3.4 Application of loads

Depending on the hardness of foods, the average bite force reportedly ranges from 20 to 120 N\textsuperscript{112}. In the present study, we applied a static load of 100 N to simulate loading by occlusion as reported in some previous studies.\textsuperscript{85,113} The vertical loads were applied in two different locations, one along the long axis of the implant at the central fossa and the 2\textsuperscript{nd} load on the tip of the buccal cusp. An oblique load was applied only on the tip of the buccal cusp. This was done to simulate and realistic chewing pattern. For a direct and systematic comparison, the same loading conditions, boundary conditions and constraints were applied in all the models.

3.5 Von Mises stress

In FEA, the mechanical performance of the implant–abutment interface could be evaluated by Von Mises stress. Von Misses stress criterion is important to interpret the stresses within the ductile material, such as the implant material, as deformation occurs when the Von Mises stress value exceeds the yield strength.\textsuperscript{86} One of the most easy way to check when a material fails is a simple tension test. Here the material is pulled from both ends. When the material reaches the yield point (for ductile material) the material can be considered as failed. The simple tension test is a uni-directional test, as shown in the Fig.3.7.
If we consider a situation like in Fig. 3.8, an actual engineering problem with a complex loading condition, we may assume that the material fails when the maximum normal stress value induced in the material is more than the yield point. If such assumptions are made, it will be based on a failure theory called 'normal stress theory'. Many years of engineering experience has shown that normal stress theory doesn’t work in most of the cases. The most preferred failure theory used in industry is ‘Von Mises stresses based. The concept of Von Mises stress arises from the distortion energy failure theory which is comparison between 2 kinds of energies, 1) Distortion energy in the actual case 2) Distortion energy in a simple tension case at the time of failure. According to this theory, failure occurs when the distortion energy in actual case is more than the distortion energy in a simple tension case at the time of failure.