Chapter 6

Conclusion
CHAPTER 6
CONCLUSIONS

1. All the four implant abutment designs studied showed the least amount of stress on the various areas of the assembly and the surrounding bone, when the loads applied were close to the long axis of the implant.

2. Both the single and two piece conical connections dissipated the least amount of stress to the abutment screw and the surrounding bone, as most of the stresses were recorded at the implant abutment interface.

3. The parallel hex connection was the least favorable design as it showed a great amount of stress on the abutment screw on vertical loading and a higher amount of stress on the bone on vertical or oblique loading of the slopes of the buccal cusp.

4. The tri-channel cylindrical connection design was stable on loads applied along and closer to the implant axis. But produced high amount of stress on the bone on oblique loads applied at a distance from the long axis of the implant.

5. The new design that placed the connection more apical to the marginal bone significantly reduced the stresses recorded at the crestal bone, but the stresses on the implant abutment interface and the abutment screw were significantly very high.

6.1 Highlights of the study:

This study was taken up with the aim of analyzing the stress distribution pattern around the existing IAI designs and to design a new implant abutment connection to reduce the stress on the bone surrounding the implant.

The designing of an internal conical interface with a tri channel anti rotational indexing feature on the superior aspect of the connection interface, have seen to dramatically enhance the ability of the IAI to minimize the stress concentration on the bone surrounding the bone to a significant degree. It must be emphasized that the stresses recorded were much lower than the stresses generated by any of the existing designs that were included in the study. It is also very critical to highlight that this is
even true in case of the oblique loading conditions. Such improved biological stability may have ramifications on the clinical integrity of an implant supported prosthesis.

6.2 Limitation of the study:

It has been observed that the three dimensional FEA was an effective means for investigating the behavioral tendencies of implants under loading conditions. However the results need to be read cautiously with the full understanding that FEA results are a purely numerical data that are mathematically enumerated.60 It must also be considered that this was not an in vitro test and cyclic loading was not examined. It is well accepted that in vivo performance does indeed differ from a FEA or an in vitro setting. There are certain limitations in this study some of which are universal to all FEAs and others which may be managed more appropriately such as following assumptions that were made in the present simulation a) the interface between the implant and the cortical or cancellous bone were completely bonded which may not be the case in clinical conditions. b) all the implant abutment connection are assumed to be bonded with each other which may not be the exact simulation of the clinical condition. c) loading these models only at a 45 degree angle reproduces only one of many possible mechanical conditions of the oral cavity.

6.3 Scope for future work.

Further modifications to this design will be done in order to minimize the stresses on the implant fixture, implant abutment interface and the abutment screw, without affecting the stress levels on the bone. The design will be modelled and analyzed using FEA, based on the data obtained a laboratory model will be fabricated and tested in vitro, to establish the strength and tolerance of the new design. A final design will then be manufactured and tested in vivo.