Optimization of the Conical Angle Design in Conical Implant–Abutment Connections: A Pilot Study Based on the Finite Element Method

Kuang-Ta Yao, DDS, PhD
Chen-Sheng Chen, PhD
Cheng-Kung Cheng, PhD
Hsu-Wei Fang, PhD
Chang-Hung Huang, PhD
Hung-Chan Kao, PhD
Ming-Lun Hsu, DrMedDent

Conical implant–abutment connections are popular for their excellent connection stability, which is attributable to frictional resistance in the connection. However, conical angles, the inherent design parameter of conical connections, exert opposing effects on 2 influencing factors of the connection stability: frictional resistance and abutment rigidity. This pilot study employed an optimization approach through the finite element method to obtain an optimal conical angle for the highest connection stability in an Ankylos-based conical connection system. A nonlinear 3-dimensional finite element parametric model was developed according to the geometry of the Ankylos system (conical half angle $\alpha = 5.7^\circ$) by using the ANSYS 11.0 software. Optimization algorithms were conducted to obtain the optimal conical half angle and achieve the minimal value of maximum von Mises stress in the abutment, which represents the highest connection stability. The optimal conical half angle obtained was $10.1^\circ$. Compared with the original design ($5.7^\circ$), the optimal design demonstrated an increased rigidity of abutment (36.4%) and implant (25.5%), a decreased microgap at the implant–abutment interface (62.3%), a decreased contact pressure (37.9%) with a more uniform stress distribution in the connection, and a decreased stress in the cortical bone (4.5%). In conclusion, the methodology of design optimization to determine the optimal conical angle of the Ankylos-based system is feasible. Because of the heterogeneity of different systems, more studies should be conducted to define the optimal conical angle in various conical connection designs.

Key Words: conical implant–abutment connection, conical angle, nonlinear finite element analysis, design optimization, Ankylos implant system, abutment fracture

INTRODUCTION

In recent years, the use of dental implants has become a reliable treatment modality for single-tooth restorations because of the well-documented high success rate of osseointegration. Nevertheless, the longevity of implant therapy remains a critical concern. Biological and mechanical complications, such as crestal bone resorption and screw loosening, are particularly problematic. Both are related to the connection. To be more specific, there is an inevitable microgap in the connection between an implant and an abutment, which tends to result in bacterial accumulation and stress concentration. A direct correlation between misfit and joint instability was proven, and the misfit should be minimized.

Conical implant–abutment connections have been developed and have become popular because of the perfect stability that is achieved through implant–abutment connections, which reduces the incidence of the aforementioned complications. The efficient clinical performance of conical connections is attributable to their large clamping force, which is transformed from the large frictional resistance in the conical interface and helps 2-piece connections function as a single entity.

In terms of the conical connection mechanism, frictional resistance originates from the geometric characteristic of the cone in the connection, allowing the abutment to sink into the implant bore. Theoretically, the degree of the conical angle can
directly control the amount of frictional resistance. A smaller conical angle leads to greater frictional resistance, and consequently benefits the connection stability. However, a smaller conical angle also results in reduced abutment rigidity, which compromises the connection stability. In other words, the conical angle exerts opposing effects on 2 influencing factors crucial to connection stability: frictional resistance and abutment rigidity. Clinically, the conical angles vary in different conical connection implant systems, and little information on this topic is available.

With increasing significance, several clinical failure cases have reported that all abutment fractures of the Ankylos implant system (Dentsply-Friadent GmbH, Mannheim, Germany), a reliable conical implant–abutment connection system, occurred horizontally at the implant platform level (Figure 1a). This was originally considered an uncomplicated clinical problem solvable by exchanging a new abutment. However, the large clamping force from its distinctive frictional resistance in the connection made retrieving the fracture abutment difficult, thereby the aggressive treatment of implant body removal with trephine as a final solution. Accordingly, we analyzed these failed retrieval cases to investigate the possible causes of failure and the influencing factors.

One of the common characteristics of these cases was that all the failures occurred in posterior single implant restorations after 1 to 2 years of service. In addition, scanning electron microscopy observations of the fracture surface demonstrated that the end of the fracture was not exactly opposite the fracture origin and that the directions of the crack propagation were multiple. These findings indicated that the abutment withstood not only bending forces but also torsional forces.
microscopy observations of fractured surfaces demonstrated that the failure patterns had the characteristics of fatigue in ductile materials under not only the bending forces but also the torsional forces originating from chewing (Figure 1c). From these observations, it can be assumed that the actual cause of these failures is weakness in the rigidity of the abutment, which is thus unable to withstand the bending and torsional forces in the posterior areas. Our findings are consistent with a finite element analysis (FEA) study warning that the reduced abutment diameter of the Ankylos system, originating from its small conical angle, may increase the risk of abutment fracture.16 Elsewhere, a Korean clinical study demonstrated a similar concern, reporting that the incidence of abutment fracture of the Ankylos implant system in the Korean population was up to 2.2%.17 In addition, difficulty in retrieval was encountered. In brief, the conical angle in this system, simultaneously controlling the abutment rigidity and frictional resistance with opposing effects on the connection stability, appears to overemphasize the frictional resistance and neglect the abutment rigidity. Consequently, this leads to the difficult clinical situation in which the abutments fracture and the fracture fragments are too tight to be retrieved.

From a mechanical engineering perspective, directly increasing the conical angle to increase the abutment diameter and thereby increase abutment rigidity appears to be a feasible solution to mitigate this retrieval problem. However, because occlusal loading is exerted on the whole conical implant–abutment connection rather than on the abutment alone, increasing the abutment rigidity is not sufficient to prevent abutment fracture; instead, the rigidity of the whole connection (ie, the connection stability), should be increased. The higher the connection stability, the smaller and more favorably distributed the stress in the abutment is, which thus contributes to a lower abutment fracture possibility. In sum, the stress condition in the abutment is a valid indicator of the connection stability.

Because the conical implant–abutment connection comprises the abutment, corresponding implant wall confining the abutment, and clamping force (frictional resistance) within the connection, the rigidity of the corresponding implant wall also affects the connection stability. Furthermore, changes in the conical angle cause different changes in the aforementioned factors of connection stability and differently affect connection stability. Increasing the conical angle increases the abutment diameter and, consequently, the connection stability. However, the thickness of the corresponding implant wall decreases as a result of the increased abutment diameter, and the clamping force within the conical connection also decreases as a result of the increase in conical angle.12,13 Both of these effects reduce the connection stability. Therefore, from an engineering design perspective, an optimal conical angle should be determined by mediating these 3 factors. Using a scientific methodology, the present study calculated this optimal conical angle.

Computer technology has contributed to product design, analysis, and manufacturing. First, computer-aided engineering, the use of computer software to improve product quality and durability, is widely applied. One key element of computer-aided engineering, FEA, is mostly used to evaluate and refine product designs through computer simulations rather than physical prototype testing, thus saving time, efforts, and money.18–20 Second, digital scanning and computer-aided design (CAD) technologies can be applied to the construction of simulation models.21–23 Third, computer-aided manufacturing (CAM) technology can be used to produce solid models. Currently, digital scanning, CAD technologies, and CAM technologies are also used to fabricate prostheses in clinical dental practice.

A parametric design optimization approach through FEA can be used to obtain the optimal parameter by using a series of optimization iterations. Furthermore, FEA is an effective simulation tool for solving design challenges without arduous manual iterations or prototyping. The ANSYS FEA package (ANSYS Inc, Canonsburg, Pa) has a powerful design optimization module in structural design optimizations.24 In addition, ANSYS Parametric Design Language, a scripting language that can be used to build a model in terms of variables, was used to build a model parametrically to enable variable changes during the optimization process. This process systematically and efficiently adjusts the influencing parameters to determine the solution with the optimal performance, satisfying given constraints. The optimization can be represented by the following mathematical mode:25

\[
\text{Minimize } f(x) \\
\text{Subject to } s_i(x) \leq S_i, \quad i = 1, 2, \ldots, m \\
X_j^L \leq x_j \leq X_j^U, \quad j = 1, 2, \ldots, n
\]

where \(f(x)\) is the objective function of independent variable \(x\) and represents the best performance that must be achieved. In a general optimization problem, the objective is to minimize the objective function. However, if the aim is to maximize the objective function, a new objective function can be set to take the negative number or the reciprocal of the original objective function.

In the equation, \(x\) denotes the independent variables in the design optimization, known as design variables, which indicate that certain parameters must be modified or adjusted. These design variables are subject to lower and upper limits, \(X^L\) and \(X^U\), respectively.

In addition, \(s(x)\) denotes the state variables that change during optimization processes depending on the design variables. The state variables indicate the reactions of the test structure after loading and are also bounded by lower and upper limits, \(S^L\) and \(S^U\), respectively.

By using design optimization, this FEA study obtained a specific conical angle that minimizes abutment stress and represents the highest connection stability for the lowest abutment fracture possibility in the Ankylos-based conical implant–abutment connection system. In addition, the optimal conical angle of the optimal design was compared with that of the original design to determine the extent of improvement.

**Materials and Methods**

**Finite element models**

**Finite Element Model Design**

In this study, we used ANSYS 11.0 to develop the finite element (FE) model and to perform FE analyses and optimization
iterations. The 3-dimensional FE model comprised a superstructure, abutment, implant and bony block. In the aforementioned clinical failure cases, the geometries of an Ankylos B11 implant (diameter: 4.5mm; length: 11mm) and a standard C/abutment (b/1.5/6.0 straight) were used as references for FE models of the implant and abutment parts. In particular, because the aim of this study was to obtain the optimal conical angle through optimization iterations, we set the conical half angle $\theta$ as the design variable. Other related conical connection parts were built according to the variation of this variable. A $\theta$ of $5.7^\circ$ represented the original design of the Ankylos implant system (Figure 2). In addition, to simplify the modeling, the threads of the implant body and abutment screw were not represented as spirals but as symmetrical rings.26,27

For FE model validation with in vitro tests, the geometries of the bony block and superstructure were modeled using cuboid blocks, which allowed a firm fixation of samples and a stable loading point in the experimental settings.28 The bony block was $17 \times 17 \times 15$ mm$^3$ in size and included a 1.5-mm cortical layer. The superstructure was $11 \times 11 \times 10$ mm$^3$ in size with a $60^\circ$ inclined plane on the top for fulfilling the following loading conditions.

The FE model was developed using SOLID187 with 10-node tetrahedral elements that are suitable for developing meshes on irregular bodies.28 In total, the FE model comprised 603,406 nodes and 386,044 elements (Figure 3).

### Material Properties

All materials were assumed to be linearly elastic, homogeneous, and isotropic. The implant was made of titanium, and the superstructure and abutment were made of a titanium alloy. Poly(methyl methacrylate) (PMMA) and stainless steel were used to replace the bone to validate the FE models in the corresponding in vitro test. The properties of the materials are listed in Table 1.

### Interface Conditions

The bone–implant interface (which simulates 100% osseointegration), superstructure–abutment interface, and cortical– cancellous bone interface were assumed to be completely bonded.
Axial displacement and the data were applied to the formula manufacturer, was measured in vitro (averaging 25
the application of 25 Ncm torque, as recommended by the
interference value was approximately 2.5
between the abutment and implant in the conical
the interference in the implant abutment connection, defined by \( \theta = \Delta z \times \tan \theta \) (where \( \theta \) is the interference value, \( \Delta z \) is the axial
displacement, and \( \theta \) is the conical half angle).

To simulate real behavior at the abutment and implant
interface, an interference value should be imported into FE
models. In other words, at the same level, the dimension of the
abutment should be slightly larger than that of the implant
bore by the interference value. This results in overlapping of
the contact boundaries. However, in reality, the interference
value (\( \delta \)) is too small to be directly measured; therefore, it is
calculated using the formula \( \delta = \Delta z \times \tan \theta \), where \( \Delta z \) is the axial
 displacement and \( \theta \) is the conical half angle (Figure 4).\(^{13}\)

In our study, axial displacement (\( \Delta z \)) of the abutments after
the application of 25 Ncm torque, as recommended by the
manufacturer, was measured in vitro (averaging 25 \( \mu m \)), and
the data were applied to the formula \( \delta = 25 \times \tan (5.7^\circ) \). The
interference value was approximately 2.5 \( \mu m \) and was imported
into the FE models. Nonlinear contact with friction was
assumed between the abutment and implant in the conical
interface, and the abutment–implant conical contact was
modeled using elastic surface-to-surface contact elements
(conta174 and targe170). The friction coefficient (\( \mu \)) adopted
for the conical interface was 0.3.\(^{16}\)

**Loading and Boundary Conditions**

To simulate the conditions of human mastication in the
posterior area, a 30\(^\circ\) off-axis loading of 200 N was applied
eccentrically 4 mm to the right of the center of the
superstructure and 10.5 mm above the platform of the implant
(Figure 5a).\(^{36}\) The generated axial loading, bending moment,
and counterclockwise torsional moment was 173.2 N, 105 Ncm,
and 40 Ncm, respectively. Additionally, the models were
constrained in all directions at the nodes on the mesial, distal,
and lower bone surfaces (Figure 5b).

**FE model validation**

In the experimental test, each implant was embedded in PMMA
(Hygenic Repair Acrylic, Coltene/Whaledent, Langenau, Ger-
many) and confined to the center of a stainless steel form.\(^{37}\)
The corresponding abutment was connected to the implant
and tightened at 25 Ncm. A titanium superstructure was then
cemented onto the abutment with resin-modified glass
ionomer luting cement (Relyx Luting 2 cement; 3M ESPE, St
Paul, Minn). The implant–abutment assemblies were consistent
with those used in the aforementioned FE models. The titanium
superstructure and stainless steel form corresponding to the
bone block in the FE models in this study were developed using
CAD/CAM according to the dimensions of the FE models.

A specimen was subjected to a 90\(^\circ\) off-axis load on the
lateral surface of the superstructure, 10.5 mm above the
platform of the implant (Figure 6a). The load test was
performed using a universal testing machine (Electric
3200; Bose, Eden Prairie, Minn). A 1-N preload was applied prior
to the test load from 0 to 155 N at a crosshead speed of 15 N/s.
Load–displacement curves were recorded, and the stiffness was
calculated, followed by the calculation of the displacement
value under 100 N.

For validation, the FE models were slightly modified by
changing the loading condition and material properties of the
PMMA and stainless steel corresponding to the experimental
test (Figure 6b). The loading magnitude was set to 100 N, and
the displacement of the loading point was recorded. A
comparison of the experimental outcomes was performed to
validate the accuracy of the models.

**Optimization analysis**

To obtain the highest connection stability for the lowest
abutment fracture possibility, an objective function \( f(x) \) was set
to determine the minimal abutment stress. In this study, the
stress status of the abutment was represented by the maximum
von Mises stress in the abutment, which was defined as the
state variable.

The design variable \( \theta \) was the conical half angle. By fixing
the bottom radii of the conical portion of the abutment, the
entire conical portion increases as \( \theta \) increases, and the
thickness of the corresponding implant wall consequently
decreases; moreover, \( \theta \) was set to a range between 1.5\(^{\circ}\) and
16\(^{\circ}\). Notably, an initial value must be applied for an initial
analysis before running the optimization algorithm. Based on
the original design of the Ankylos system, the initial value of \( \theta \)
was set to 5.7\(^{\circ}\) (Figure 2).

In every optimization iteration, every state variable
obtained, depending on its corresponding design variable,
was the maximum von Mises stress in the abutment. To ensure
the rigidity of the abutment and implant, the boundary of the
state variable was set to below the ultimate tensile strength of
the titanium alloy (910 MPa).\(^{38}\) Therefore, the objective function
was to obtain the minimized state variable.

This study performed an optimization algorithm by using
the subproblem approximation method. This method can be
described as an advanced, zero-order method. A convergence
tolerance of 0.001 was provided in the program.

**Comparison of the optimal and original designs**

The optimal design was compared with the original design in
terms of several postprocessing FE results to establish the
extent of improvement. First, the maximum von Mises stress values in the abutment and implant were examined to elucidate their abilities to resist structure fracture. Second, the maximum values and the distribution of the contact pressure in the connection were examined to represent the conditions of the clamping force in the connection. Then, the maximum values of the microgap at the implant–abutment interface were examined, which could be indices for the performance of the connection stability during loading and the health of tissues around the connection. Finally, the maximum values of the principal stress of the cortical bone were examined to elucidate the stress status of the surrounding bone.

**RESULTS**

**FE model validation**

The experimentally measured and numerically calculated displacement values at the same loading point were 0.33 and 0.28 mm, respectively, indicating a reasonable model validation.

According to the results of the initial value (5.7°), the maximum von Mises stress occurred on the abutment at the implant platform level, consistent with the abutment fracture area observed clinically (Figure 1a and b). These corresponding findings can also be used for FE model validation.

**Optimization analysis**

This optimization process comprised 5 iterations, each of which is shown in Figure 7. The optimal conical half angle was calculated to be 10.1°.

**Comparison of the optimal and original designs**

A comparison of the original and optimal designs is presented in Table 2. Notably, the optimal design exhibited reduced maximum von Mises stress in both the abutment and implant by 36.4% and 25.5%, respectively. Because the maximum von Mises stress of the abutment was reduced from 683.70 to 434.73 MPa, the abutment of the optimal design significantly reduced fracture probability. The same positive effect occurred in the implant of the optimal design.

In terms of clamping force in the connection, although the maximum contact pressure in the connection of the optimal design was reduced by 37.9%, a more uniform stress distribution was observed (Figure 8). The maximum microgap at the implant–abutment interface of the optimal design was reduced by 62.3%, indicating significant improvement in the connection stability. The reduced microgap also contributes to the health of the surrounding tissue. In addition, the maximum principal stress of the cortical bone in the optimal design was reduced by 4.5%, indicating that the optimal design is beneficial to the maintenance of the surrounding cortical bone.

**DISCUSSION**

In this study, 3-dimensional FE models were experimentally validated before optimization processes were conducted. Because the displacement value measured in the experimental test was very close to the value of the FEA estimate, the accuracy and rationality of these FE models was confirmed. Nevertheless, the displacement value calculated from the FE models was lower than the actual experimentally measured value. The cause of this negative deviation may have been the discrepancy between the perfect assumption of linear elasticity, homogeneity, and isotropism in the FE models and the imperfect conditions in the setup of the experimental tests due to experimental errors.

In terms of FE modeling, the interference phenomenon (an important characteristic of conical connections) was simulated and imported into the FE models; notably, in most FEA studies, this phenomenon has been neglected. In addition, the use of simplifying threads on the implant body and the abutment screw in the models to conserve computational resources in the execution of optimization calculations and to facilitate mesh construction was proved feasible by the positive validation result.

The conical angle was considered to be the key parameter influencing the dynamic stability of the conical connections. In the analysis of the clinical failure cases, the conical angle exerted opposing effects on abutment rigidity and frictional resistance in the connection. Therefore, in this study, the conical angle was set as the design variable to determine the value that would create the highest connection stability. In addition to the conical angle, the mechanical behaviors of conical connections were determined using material properties, the coefficient of friction, the depth of insertion (interference), and geometric factors that include contact length and inner and outer diameters of the members. Because of the heterogeneity of different implant systems, not all of these parameters in every implant system are consistent; we therefore used the Ankylos implant system analysis cases as a reference in this pilot study to control them. We suggest that additional studies on different conical connection systems

---

**TABLE 1**

Mechanical Properties used in the finite element analyses

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s Modulus (E) (MPa)</th>
<th>Poisson’s Ratio (ν)</th>
<th>References</th>
</tr>
</thead>
<tbody>
<tr>
<td>Titanium</td>
<td>110 000</td>
<td>0.35</td>
<td>Benzing et al, 29</td>
</tr>
<tr>
<td>Titanium alloy (Ti-6Al-4V)</td>
<td>110 000</td>
<td>0.35</td>
<td>Chang et al, 10</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>13 700</td>
<td>0.3</td>
<td>Saidin et al, 31</td>
</tr>
<tr>
<td>Cancellous bone</td>
<td>1370</td>
<td>0.3</td>
<td>Pessoa et al, 12</td>
</tr>
<tr>
<td>Acrylic resin (poly[methyl methacrylate])</td>
<td>1500</td>
<td>0.35</td>
<td>Pessoa et al, 34</td>
</tr>
<tr>
<td>Stainless steel</td>
<td>200 000</td>
<td>0.31</td>
<td>Yaman et al, 35</td>
</tr>
</tbody>
</table>
Figures 5 and 6. Figure 5. Loading and boundary conditions in the finite element analysis. (a) Loading conditions in the optimization analysis simulating posterior occlusion in the oral cavity. (b) Boundary conditions. Figure 6. Finite element model validation. (a) Experimental test setup for validation. (b) Loading conditions corresponding to the experimental test in the finite element model for validation.
Mises stress of the abutment was minimal (434.73 MPa) and the implant (25.5%), a lower microgap at the implant–abutment interface (62.3%) and a lower maximum principle stress in the cortical bone (4.5%). In sum, the substantial improvement in all comparisons indicated that this optimization study had an important clinical value. Therefore, future research should focus on making a real product based on the optimization design and comparing it with the original product to prove its value in clinical applications.

The relationship between connection stability and the conical angle in the conical implant–abutment connection is presented in the line graph in Figure 7, in which the relationship between the conical half angle and maximum von Mises stress in the abutment (a valid indicator of connection stability) was predicted through the optimization process. Two stages with opposing trends were observed. At the first stage, when the conical half angle was below 10.1°, the connection stability increased with an increase in conical half angle. This is possibly attributable to the fact that the stiffer abutment helped overcome the negative effects of the decreased clamping force and the decreased implant rigidity. In other words, in terms of overall connection stability, the role of abutment rigidity was more critical than that of implant rigidity and clamping force. At the second stage, when the conical half angle was above 10.1°, the connection stability decreased with an increase in the conical half angle. This was possibly a result of both the abutment rigidity and conical half angle increasing, which prevented the connection from overcoming the negative effects of the decreased clamping force and implant rigidity, and thus resulted in a decreased connection stability.

The magnitude of stress and its distribution were considerably influenced by loading conditions in the FE analysis; therefore, the setting of loading conditions was an essential factor in this study. Based on scanning electron microscopy observations of clinical failures (Figure 1c), the end of the fracture was not exactly opposite the fracture origin, and the directions of the crack propagation were multiple. These findings indicated that the abutment withstood not only bending forces but also torsional forces, consistent with chewing patterns in the posterior area.36,40 In addition, failure was observed in all the posterior restoration cases. Therefore, the loading condition, comprising force magnitude and force moments, was set in accordance with the posterior occlusion in the oral cavity in this study.

From a mechanical engineering perspective, the clamping force in conical connections originates from a large frictional resistance. Because the diameter of the male cone is slightly

![Figure 7](image-url)

**Figure 7.** Line graph composed of all optimization iterations (blue point), which are also listed in the lower table. The graph reveals the relationship between the maximum von Mises stress in the abutment and conical half angle. The asterisk indicates the optimal conical half angle (10.1°) at which the value of the maximum von Mises stress of the abutment was minimal (434.73 MPa).

Table 2

<table>
<thead>
<tr>
<th>Original Design</th>
<th>Optimal Design</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conical half-angle</td>
<td>(θ = 5.7°)</td>
<td>(θ = 10.1°)</td>
</tr>
<tr>
<td>Maximum von Mises stress of abutment</td>
<td>683.70 MPa</td>
<td>434.73 MPa</td>
</tr>
<tr>
<td>Maximum von Mises stress of implant</td>
<td>504.81 MPa</td>
<td>376.26 MPa</td>
</tr>
<tr>
<td>Maximum principal stress of cortical bone</td>
<td>36.60 MPa</td>
<td>34.97 MPa</td>
</tr>
<tr>
<td>Maximum microgap between implant-abutment connection</td>
<td>2.75 µm</td>
<td>1.04 µm</td>
</tr>
<tr>
<td>Maximum contact pressure in the connection</td>
<td>514.20 MPa</td>
<td>319.33 MPa</td>
</tr>
</tbody>
</table>
larger than that of the female receptacle at the same level, a wedge effect occurs. Contact pressure is subsequently generated on both surfaces when the male cone is fitted into the female element. Therefore, the discrepancy between the abutment and implant bore at the same level, defined as interference \( \delta \), is characteristic of conical connections.

To simulate real behavior, an interference value and its contact elements were imported into the FE models. Because of this phenomenon, contact pressure was generated in the connection while screwing and was transformed into a clamping force that stabilized the connection prior to occlusal loading. However, the contact pressure within the connection led to internal stressing of the abutment and implant, which increased with a decrease in the conical angle. In the original design, a smaller conical angle resulted in the storage of a larger contact pressure in the connection. Therefore, under the same loading conditions, the stresses on the implant and abutment were higher in the original design compared with the optimal design. Consequently, a larger clamping force in the connection is not necessarily more effective than a smaller one. In fact, the optimal value is the appropriate value, not the largest value. An appropriate clamping force in the connection contributes to low stress on implant components and facilitates retrieval in case of abutment fracture.

In engineering, design optimization is an advanced concept because optimization requires tedious mathematical operations. An FE model combined with optimization analysis is an effective simulation tool for facilitating the design of medical devices, such as spinal cages, thumb spica splints, spinal braces, and implants. In this study, this methodology helped improve a current implant product on the market by enhancing its durability. Through an analysis of failed retrieval cases, this study can provide researchers with in-depth information to determine the possible causes of failure. The conical angle was set as a design variable to be optimized. By using design optimization, the optimal angle was calculated using the conditions of the objective demands and material constraints, which is more convincing compared with the determination by skilled designers based on their knowledge, experience, and judgment. Furthermore, manufacturers of individual implant systems can continuously improve their systems’ performance according to specific clinical problems by using the method in this study.

**CONCLUSION**

Within the limitations of this study, the optimal design of the Ankylos-based implant system (conical half angle = 10.1°) was determined using design optimization, which proved to not only reduce the possibility of abutment fracture but also increase the longevity of the implant therapy overall. Because of the heterogeneity of different systems, we suggest that additional studies be conducted to define the optimal conical angle in various conical connection designs.

**ABBREVIATION**

CAD: computer-aided design  
CAM: computer-aided manufacturing  
FE: finite element  
FEA: finite element analysis  
PMMA: poly(methyl methacrylate)

**ACKNOWLEDGMENT**

This study was supported by a research grant (MOST 103-2314-B-010-023) from the Ministry of Science and Technology in Taiwan.

**NOTE**

The authors declare that they have no conflicts of interest.

**REFERENCES**

1. Romeo E, Lops D, Margutti E, Ghisolfi M, Chiapasco M, Vogel G. Long-term survival and success of oral implants in the treatment of full and...