Progressive failure analysis of laminated composite femoral prostheses for total hip arthroplasty

Chaodi Li\textsuperscript{a}, Christopher Granger\textsuperscript{b}, H. Del Schutte Jr.\textsuperscript{c}, Sherrill B. Biggers Jr.\textsuperscript{b}, John M. Kennedy\textsuperscript{b}, Robert A. Latour Jr.\textsuperscript{a,}\textsuperscript{*}

\textsuperscript{a}Department of Bioengineering, Clemson University, 501 Rhodes Engineering Research Centre, Clemson, SC 29634, USA
\textsuperscript{b}Department of Mechanical Engineering, Clemson University, Clemson, SC 29634, USA
\textsuperscript{c}Department of Orthopaedic Surgery, Medical University of South Carolina, Charleston, SC 29425, USA

Received 1 June 2001; accepted 11 May 2002

Abstract

In this research program, a numerical method was developed to predict the progressive failure of a thick laminated composite femoral component for total hip arthroplasty. A 3-D global/local technique was used to capture the overall structural response of this system while also enabling the 3-D ply level stress state to be determined efficiently and accurately. Different failure criteria and different material degradation models were incorporated as individual subroutines in the numerical method, giving it the flexibility to model a wide range of materials and structures. Numerical modeling was also conducted to design experimental test methods for component fatigue testing that closely simulate in vivo loading conditions. Parametric studies were then conducted with the numerical model of the experimental system and the results were compared to the actual experimentally determined damage behavior of fabricated laminated composite femoral component to assess which parameter set most accurately predicted the actual damage development behavior. The best fitting parameter set was then applied to the failure problem of the composite hip prosthesis implanted in an anatomically modeled femur to predict in vivo performance. This work provides a ply level understanding of the damage behavior of laminated composite femoral components and a numerical tool which can serve as a guide for the design of fatigue resistant implants made from composite material for this and other implant applications. © 2002 Elsevier Science Ltd. All rights reserved.

Keywords: Failure; Composite; Prosthesis; Finite element; Hip replacement

1. Introduction

It has been estimated that over 800,000 total hip replacements are performed worldwide annually [1,2]. A major cause of the failure of non-cemented metallic femoral components used for hip joint replacement is component loosening. While it is still argued as to which factors are most responsible for this problem, one of the contributing factors is recognized to be stress shielding induced osteopenia. Proximal bone loss also makes revision surgery more complicated by compromising the bony foundation for the new femoral component [1–4]. Advanced composite materials with their tailorability in stiffness and strength are potentially good materials for hip prostheses applications. Composite’s inherent tailorability allows creation of modular stiffness design and could potentially result in the reduction of stress shielding and subsequent resorption in the proximal femur [4–10]. However, the more flexible the stem, the higher the proximal stem–bone interface stresses [3]. These interface stresses may cause proximal stem debonding and relative micro-motions at the interface. Clearly, it is necessary to satisfy a number of design requirements. With a laminated composite stem, the designer has the freedom to vary the orientation of each ply and the stacking sequence to achieve beneficial stiffness, stress distributions, component strength, and physiological performance [4,9]. With this in mind, multi-level optimization methods have been developed for designing composite stems with the objective to minimize bone stress shielding effects while keeping the bone/stem interface stresses below user-defined
maximum levels [6]. As with all hip prostheses, initial fixation and stability are of concern. Achieving good initial stability is extremely important and preventing long-term prosthesis migration is critical for successful implant design. Most of the composite hip stem designs under consideration are of the cementless type that incorporate some form of fixation into their design, such as press-fit. Studies have shown that the press-fit composite hip stem compared quite well with conventional metallic press-fit hip stems in considering the axial migration and stem subsidence [8]. Research has also shown that with appropriate stem design and insertion technique, good initial stability can be obtained with a composite hip stem [8]. Due to their potential to better maintain bone stock, research on advanced composite materials for hip prostheses applications continues. Animal studies showed that the use of low stiffness composite stems can enhance proximal bony ingrowth, increase proximal medullary bone density, reduce proximal cortical bone loss, and prevent distal hypertrophy [4,7]. Although the potential for improved performance exists, clinical studies reported early fatigue fracture of a femoral component made from laminated fiber reinforced polymer composites [7]. Thus, hip stem strength is a serious concern. Therefore, before the potential of composite materials for application to total hip arthroplasty (THA) in humans can be realized, further analysis and design studies are obviously needed to improve their damage resistance to a level necessary for this application.

Progressive failure analysis on composite implants can provide important information on the damage initiation and accumulation process. However, the complex structure (geometry and the material distributions) of the femur and the loading conditions that need to be considered in such an analysis present a rather daunting task requiring fully 3-D stress analysis [9–11]. Furthermore, because laminated composite femoral components are thick composite structures typically containing more than 100 plies, laminated composites in THA applications represent a highly complex multiscale structural design problem [9–11]. Computation times needed to conduct a standard finite element (FE) analysis to evaluate the stress state of each individual ply within the prosthesis are prohibitory long, even for the most powerful computer systems available today [9–14]. It is still a very challenging problem to obtain desired levels of both computational efficiency and solution accuracy using standard FE methods for stress analysis of thick laminated composite implants, let alone to predict damage initiation and propagation within composite hip implants as a function of laminate design. Because of this complexity, few studies have been published to date on the numerical failure analysis of laminated composite hip implants [12–14].

To address this problem, a progressive failure analysis methodology has been developed by the authors for the damage analysis of general thick laminated composite structures [14,15]. This methodology is an extension of previously reported 3-D global/local methods [11,16] and optimization techniques [6,17] for the accurate 3-D stress analysis and design of thick composite laminates for THA. In the developed 3-D global/local method, an adequate coarse global FE analysis of the whole femur/implant structure was first performed, and the high stress critical region was then identified from the global solution. A refined local 3-D solid FE model was then used to analyze the critical region in the structure where damage was predicted to initiate and to propagate. The goal of this analysis technique was to combine the accuracy of a full 3-D ply level stress analysis in the interest region with the computational efficiency provided by global/local analysis. Preliminary results from numerical verification studies have indicated that the 3-D global/local method can result in as much as 1–2 orders of magnitude reduction in CPU time while still maintaining a high degree of accuracy in comparison with a conventional fully refined model [11,15,16,18]. Stress analysis based on this method also has the distinct advantage that it can be used to readily locate the high-stress critical regions of the composite structures and thus does not require prior definition of the local model. Therefore, this technique is able to capture the overall structural response while enabling the 3-D ply level stress state to be determined efficiently and accurately. In the present extension of this analysis technique, different failure criteria and different material degradation models were incorporated as individual subroutines in the 3-D progressive failure analysis methodology, thus providing the flexibility to model the failure processes for a wide range of materials and structures. Numerical results have shown that this methodology correctly predicts the damage initiation and accumulations in laminated composite structures [14,15].

This research is concerned with the design of fatigue resistant composite hip prostheses. In this paper, the previously developed progressive failure analysis methodology [14,15] is applied to evaluate how ply level damage initiation and propagation is affected by variations in ply orientation and stacking sequence of a laminated composite femoral component. By understanding the relationships between implant design and their structural responses, composite femoral components can be optimized to minimize the probability of component failure, thus enabling the potential benefits offered by low-stiffness composite femoral components to be finally realized for improving the longevity and performance of hip joint replacement in patients. Alternatively, this approach also has general applicability for failure analysis of any complex hierarchical system.
2. Materials and methods

2.1. Overview

This paper presents the application of the progressive failure analysis methodology [14,15] developed for the analysis of actual laminated composite femoral components for hip joint replacements. Before this methodology could be applied to predict the damage development in composite femoral components, an appropriate failure model with an appropriate material property degradation parameter set had to be identified to calibrate the damage model to experimentally determine femoral component fatigue behavior. In order to do this, an experimental test fixture was first designed and fabricated using FE analysis to provide a stress state within the composite femoral component that closely simulates that which occurs under in vivo loading conditions. Numerical parametric analyses using different damage models (i.e. different failure criterion and material degradation model sets) were then performed using the FE model of the experimental test system to predict how damage initiation and propagation of the composite femoral component varied based on the damage model parameters. Experimental fatigue tests were then performed in parallel to the numerical analyses to observe the actual damage development behavior in the experimental fixture [19–21]. By comparing the numerical predictions with the experimental results, a damage model parameter set was identified which predicted the damage behavior most closely in agreement with the experimental data. This calibrated damage model was then applied to analyze the damage behavior of a simulated in situ laminated composite femoral component under simulated in vivo loading conditions.

The application of the failure analysis methodology for the damage analysis of an in situ laminated composite femoral component was approached by a seven-step analysis procedure:

1. **FE modeling of femur/implant structure:** Construct an anatomical 3-D FE model of the femur/implant structure.
2. **Global analysis of femur/implant model:** Perform global FE analysis of the in situ femoral component.
3. **Experimental system design and global analysis:** Design an experimental test fixture that closely simulated the mechanical characteristics of the in situ composite component based on step 2.
4. **Global/local analysis of FE model of the experimental system:** Identify and isolate the critical stress volume in the composite component of the experimental system for detailed ply level failure analysis.
5. **Experimental testing of laminated composite femoral components:** Test, monitor and record the damage initiation and propagation of the composite stems using the fixture designed in step 3.
6. **Progressive failure analysis of FE model of experimental system:** Study the effects of different damage parameter sets on the progressive failure predictions of the FE model of the experimental system, and calibrate the FE model to simulate actual composite component fatigue behavior.
7. **Failure analysis of in situ composite femoral components:** Apply the damage model calibrated in step 6 to predict the in situ laminated composite femoral component damage development as a function of ply level design.

Each of the steps is addressed in detail in the following sections.

2.2. Step 1. FE modeling of femur/implant structure

In this research, an anatomic 3-D FE model of the right-side femur was developed from sequential transverse CT-scan sections taken every 3.0 mm along the length of an average sized adult human femur. These CT-scans provided the necessary geometric and bone density data needed for constructing an FE model of the proximal femur [11]. The anatomic 3-D femur model was constructed using the FE modeling software I-DEAS [22].

Before the femoral component could be modeled, decisions had to be made regarding its structural design. For simplicity, it was decided that the prototype composite prostheses were to be fabricated from a carbon fiber-reinforced polyetheretherketone (C/PEEK) laminated plate with the external geometry [19–21] as shown in Fig. 1; with the laminate plies lying in the $x-y$ plane. The outer geometry was selected to minimize stress concentration effects at the neck/stem junction of the prosthesis while otherwise providing a non-cemented canal-filling prosthesis sized to fit the selected femoral model. The prosthesis model was translated and rotated as necessary to attain the appropriate position when inserted into the modeled femur to construct the femur/implant solid model using FE modeling package I-DEAS.

Once the solid model was completed, the FE model was constructed using the solid model as a template. The solid model was meshed with high-order 20-node quadratic brick elements and 15-node wedge elements (element types C3D20 and C3D15 in FE software ABAQUS, Hibbit, Karlsson and Sorensen Inc., Pawtucket, RI [23]). Areas of cortical and cancellous bone within the femur were assigned different material properties based on their mineral densities as inferred from the gray scale intensity factor obtained from the CT-scan images. The cancellous bone was assigned
isotropic properties with elastic moduli varying according to bone density. Different Young’s moduli were assigned to different regions in the bone based on (1) the linear relationship between CT gray scale numbers (Houndsfield numbers) and apparent density, and (2) a non-linear cubic proportionality relationship between Young's modulus and apparent density [24]. The cortical bone was divided into metaphyseal and diaphyseal areas that were assigned to have the orthotropic properties (Table 1) as used by Cheal and co-workers [25]. The material properties of the C/PEEK composite are shown in Table 2 [26,27].

Previous studies have shown that the mechanical loading of the hip joint can be accurately represented by only applying muscle and joint forces related to designated gait activities [1]. The activity which was selected to be simulated in this work was the toe-off phase of gait, which is considered to be severe loading phase of the walking gait cycle for composite femoral components because it has large total load magnitude and large out-of-plane component of the resultant joint load vector [1,25,28,29]. Fig. 2 shows the hip joint and the involved muscle forces (adductors and abductors) for the toe-off condition [28–33]. In experiments, the force of a muscle group is typically applied with only a single wire, not with the use of multiple wires. Such a setup was selected so as to improve the control of the forces [1]. More physiological muscle actions may be achieved with the use of distribution forces applied. However, by St. Venant’s principle, the stress concentration effects will locally exist only near the muscle insertion points if the load is applied as concentrated force. Therefore, all loads were applied as concentrated loads in this paper with the loading conditions obtained from literature [28,29]. The muscle insertions on the proximal femur were defined using the appropriate nodes on the FE model with the aid of the anatomic drawings [11,30–33]. In Fig. 2, hip joint, adductors muscle and abductors muscle forces are 4.9, 0.60 and 1.79 BW, respectively. BW represents body weight, which is assumed to be about 750 N for the present analysis [25,28]. The prosthesis is considered to be collarless and press-fit (cementless type). No gap and no slippage were assumed between the prosthesis and the bone. In this research, the primary interest was the strength of the composite implants. In accordance with St. Venant’s Principle, FE analysis showed that there was little difference in stress distributions in the implant neck area (the critical stress area) when comparing bonded and frictionless unbonded interface conditions. Therefore, for simplicity, the interface between the femur and the implant was represented as being fully bonded [1]. Also, the femur was fully constrained distally [1].

### 2.3. Step 2. Global analysis of femur/implant model

In order to design an experimental fixture to represent the in vivo loading conditions experienced by a composite femoral component, the mechanical environment of an in situ femoral component first had to be simulated. An adequate global FE analysis of a femur/implant model was performed in this step to provide the

---

**Table 1**

<table>
<thead>
<tr>
<th>Cortical bone</th>
<th>Elastic modulus (GPa)</th>
<th>Poisson’s ratio</th>
<th>Shear modulus (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( E_a )</td>
<td>( E_b )</td>
<td>( E_c )</td>
</tr>
<tr>
<td>Diaphysis</td>
<td>21.9</td>
<td>14.6</td>
<td>11.6</td>
</tr>
<tr>
<td>Metaphysis</td>
<td>17.5</td>
<td>11.7</td>
<td>9.3</td>
</tr>
</tbody>
</table>

The ‘\(a\)’-axis corresponds to the long axis of the bone, the ‘\(b\)’-axis corresponds to the circumferential direction, and the ‘\(c\)’-axis corresponds to the radial direction.
At the global level, the laminated composite prosthesis was modeled using 3-D elements where each element incorporated several composite plies within it. Each layer of elements thus represented a sub-laminate in which the smeared element stiffness properties [11] were set to be equivalent to the set of plies contained within it [9,11,34–36]. This resulted in modeling efficiency by reducing the number of global elements required to capture the global structural response. The 3-D effective elastic constants of each element were derived based on the long-wave approach [34]. Mesh refinement studies were performed to determine the adequate level of discretization required to adequately represent the overall deformational response of an in situ composite hip prosthesis for the activity of toe-off. This step thus provides the overall global response in the form of the strain energy density (SED) distributions of the femur/implant structure subjected to toe-off loading conditions. This SED information was then used as the critical structural response parameter which served as the basis to design an experimental fatigue test fixture that was able to closely simulate the in vivo load conditions of an in situ composite femoral component.

The SED distribution was selected as the critical response parameter for this research based on previous FE work [9–11].

### Table 2

Material properties of CF/PEEK unidirectional composite prepreg [APC-2/AS4] [26]

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>In-plane longitudinal modulus</td>
<td>$E_{XX} = 135.3 \text{ GPa}$</td>
</tr>
<tr>
<td>In-plane transverse modulus</td>
<td>$E_{YY} = 9.0 \text{ GPa}$</td>
</tr>
<tr>
<td>Out-of-plane modulus</td>
<td>$E_{ZZ} = 9.0 \text{ GPa}$</td>
</tr>
<tr>
<td>In-plane shear modulus</td>
<td>$G_{XX} = 5.2 \text{ GPa}$</td>
</tr>
<tr>
<td>Out-of-plane shear modulus</td>
<td>$G_{YY} = 5.2 \text{ GPa}$</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>$v_{XY} = v_{XZ} = 0.34, v_{YZ} = 0.46$</td>
</tr>
<tr>
<td>Longitudinal strength</td>
<td>$X_L = 2068 \text{ MPa}, X_C = 1448 \text{ MPa}$</td>
</tr>
<tr>
<td>Transverse strength</td>
<td>$Y_L = 86 \text{ MPa}, Y_C = 250 \text{ MPa}$</td>
</tr>
<tr>
<td>Peel strength</td>
<td>$Z_L = 86 \text{ MPa}, Z_C = 250 \text{ MPa}$</td>
</tr>
<tr>
<td>In-plane shear strength</td>
<td>$S = 188 \text{ MPa}$</td>
</tr>
</tbody>
</table>

2.4. Step 3. Experiment system design and global analysis

Experiments were performed during the course of this investigation to calibrate and verify the developed numerical damage model. For this part of the research program, it was desired to conduct fatigue testing of the composite femoral components using a fatigue test method which supported and loaded the femoral component in a manner which closely simulated the mechanical environment that an in situ composite femoral component would be subjected to within a femur under loading conditions representing the toe-off phase of gait. As mentioned in the proceeding section, the SED distribution within the femoral component was selected as the most appropriate mechanical parameter to be matched for this equivalence [9–11]. In this step, a global FE model of the basic experimental setup was first developed, based partially on previously reported experimental studies used for composite femoral component evaluation [12,13]. Different test parameters (e.g., foundation design and load vector direction) were then evaluated by FE analyses and adjusted until they induced an SED distribution within the composite component which closely approximated that of the same composite component in the global femoral model.

Based on previous work for testing of a composite femoral component [12,13], an initial test fixture was designed for the fatigue testing of flexible-stemmed femoral components used in THA subjected to a periodic constant amplitude force (Fig. 3(a)) [19–21]. Detailed drawings and dimensions of the fixture were produced and translated to I-DEAS software to create a 3-D FE model for initial simulation/optimization studies. Several test fixture parameters were considered for the design process, including the fixture wall, potting materials, stem/potting material interface, stem support and stem head loading directions (Fig. 3). A dense polymer resin plastic with an elastic modulus of 3.5 GPa
CAROPLASTIC, Carolina Biological Supply Inc., Burlington, NC) was selected as the potting medium [19]. This foundation medium was found to be able to support the high load levels required for low-cycle high-load fatigue testing [19]. End-supported (i.e. the stem distal tip was supported by an end plate within the fixture) versus end-unsupported conditions (i.e. the distal tip of the stem was free) and fully bonded versus frictionless interfaces (i.e. stem/foundation interface was bonded or had a gap) were compared by FE analysis to determine the effects of these different experimental testing conditions upon the SED within the stem [19]. In the models, the resultant load vector was applied to the center of the portion of the stem at the head of the femoral component and the direction of the load acting on the head was parametrically varied to evaluate its effect on the SED distribution within the stem.

Based on the FE studies, an optimal fixture design was obtained which provided a global-level SED distribution of the composite stem in the experimental fixture which was very close in location and distribution to that of the stem in the femur under toe-off loading conditions [14]. The angles defining the line of action of the head force (see Fig. 3) were found to be $\theta_x = 85.4^\circ$, $\theta_y = 15.1^\circ$ and $\theta_z = 75.6^\circ$. $\theta_x$, $\theta_y$ and $\theta_z$ are the angles between the head force vector and the coordinate x, y and z axes, respectively, using the coordinate system defined in Fig. 1.

Following the completion of the experimental test fixture design, this research program was split into two parallel efforts: one to fabricate and fatigue test actual laminated composite femoral components and the other to numerically predict damage behavior as a function of the failure theory and damage model parameter set selected.

Two different composite femoral stems with different ply orientations and stacking sequences were designed for fatigue testing with the fabricated test fixture. The specified details of the fabrication procedures used for these composite femoral components are reported in Refs. [19–21]. These two stems were designed to have the same overall global structural stiffnesses but with different internal ply orientations and stacking sequences. This design enabled the in situ global femur/implant model to represent each of the two stems while providing ply level differences that should influence the actual stem damage behavior. The stacking sequences of these two stems (Design I and II stems in the following analysis) are shown in the following lay ups with the angles referenced to the coordinate system shown in Fig. 1 [14,19]:

I: $\{ \pm 45/\pm 60/\pm 30/\pm 60/(\pm 20)z/\pm 70/\pm 5/\pm 60/\pm 5/\pm 60/\pm 25/\pm 45/\pm 60/\pm 35/(\pm 15)x/\pm 75/\pm 30/\pm 15/\pm 25/\pm 60/\pm 55/\pm 45/\pm 20/\pm 65\}$, 

II: $[(\pm 45)z/(\pm 60)z]_{10}/[(\pm 45)z/\pm 60/\pm 35]_{c}$.

The experimental testing and numerical prediction of the laminated composite femoral components are discussed in detail in the following sections in Steps 4–7.

2.5. Step 4. Global/local analysis of FE model of experimental system

In Step 4, the global level FE analysis obtained in Step 3 for the final fixture design was utilized to generate and

Fig. 3. (a) Schematic of test fixture with hip stem, (b) final fixture design with implant.
solve the local ply level stress state for each of the femoral component designs. The global level FE solution of the composite femoral component was obtained using the developed numerical model of the experimental test fixture. The hot spot was identified using SED as the performance criterion. Once located, this hot spot area was then modeled with a fully refined ply level local model and analyzed using the previously developed progressive failure analysis methodology which was presented in Refs. [14,15]. The local domain was discretized using 20-node brick elements or 15-node wedge elements, where one brick or wedge element was used per ply in the thickness direction. Previous studies have shown that this meshing level provides reasonably accurate through-the-thickness stresses [11,37]. The boundary conditions of the local model were interpolated from the corresponding global displacement solution at the global/local interface using elemental shape functions [23], and the local model was then solved to provide the local ply level stress states. Details of the methods used for this global to local transition and the solving of the 3-D ply level stress state have been previously reported (see Refs. [11,14–16]).

2.6. Step 5. Experimental testing of laminated composite femoral components

In an experimental study conducted parallel to this numerical study, the designed test fixture was fabricated and used to conduct fatigue tests with each type of the fabricated composite femoral components (for details, see Refs. [19–21]). Damage development was monitored during the fatigue test by stereomicroscopy and X-ray radiography. These damage development results thus provided the experimental data needed to set parameters in the numerical damage development model so as to calibrate the numerical model to most accurately reflect actual stem behavior during progressive failure.

2.7. Step 6. Progressive failure analysis of local FE model of experimental system

In this portion of the research program, parametric damage model studies were conducted on local models of the femoral component extracted from the global analysis results of the experimental test fixture FE model. These parametric studies were performed to evaluate how a specific damage model parameter set influenced the predicted ply level damage development behavior in the stems. The parameters considered included three different failure criteria (Independent Criterion [33], Tsai-Wu criterion [32,33], Intralamellar and Interlamellar Criterion [9]) and two different material degradation models (total discount method and limited discount method) [36–38]. The inclusion of various failure criteria and material degradation models provided a very broad-based representation of the possible failure responses of the laminated composite structure. In other composite systems, predicted damage behavior has been previously shown to be quite sensitive to both the applied failure criteria [39] and the material degradation models [36–39]. Each failure criterion applies different weighting to the effect of each stress component with regard to its respective strength parameter, while different stiffness reduction coefficients strongly influence how stress is redistributed following element damage.

These failure criterion and material degradation models were integrated into the commercial stress analysis software ABAQUS [23]. The failure criteria are listed below:

1. Independent failure criterion:
   (a) Mode 1(Fiber failure):
   \[ |\sigma_{11}/X| \geq 1 \text{ or } \sigma_{13}\sigma_{13} + \sigma_{12}\sigma_{12} \geq \sigma_r^2, \]  
   (b) Mode 2(Transverse cracking perpendicular to 2 direction):
   \[ |\sigma_{22}/Y| \geq 1 \text{ or } \sigma_{21}\sigma_{21} + \sigma_{23}\sigma_{23} \geq \sigma_{ms}^2. \]  
   (c) Mode 3(Transverse cracking perpendicular to 3 direction):
   \[ |\sigma_{33}/Z| \geq 1 \text{ or } \sigma_{31}\sigma_{31} + \sigma_{32}\sigma_{32} \geq \sigma_{ms}^2, \]  
   where \( \sigma \) represents stress; subscript 1 represents the fiber direction, 2 represents the direction transverse to the fiber but in the plane of the laminate, 3 represents the direction transverse to the fiber and to the laminate; \( X = X_i \) if \( \sigma_{11} > 0 \); \( X = X_c \) if \( \sigma_{11} < 0 \); \( X_i \) and \( X_c \) are the tensile and compressive strength of the lamina in the fiber direction; \( Y = Y_i \) if \( \sigma_{22} > 0 \); \( Y = Y_c \) if \( \sigma_{22} < 0 \); \( Y_i \) and \( Y_c \) are the tensile and compressive strengths of the lamina in the 2 direction; \( Z = Z_i \) if \( \sigma_{33} > 0 \); \( Z = Z_c \) if \( \sigma_{33} < 0 \); \( Z_i \) and \( Z_c \) are the tensile and compressive strength of the lamina in the 3 direction; \( \sigma_{r} \) and \( \sigma_{ms} \) represent fiber shear strength and matrix shear strength, respectively [37].

2. Tsai-Wu criterion:
   \[ F_i\sigma_i + F_0\sigma_i\sigma_j \geq 1 \quad i,j = 1–6. \]  
The terms in the criterion are defined as follows:
   \[ \sigma_1 = \sigma_{11}, \quad \sigma_2 = \sigma_{22}, \quad \sigma_3 = \sigma_{33}, \]
   \[ \sigma_4 = \sigma_{23}, \quad \sigma_5 = \sigma_{13}, \quad \sigma_6 = \sigma_{12}, \]
   \[ F_1 = 1/X_T - 1/X_c, \quad F_2 = 1/Y_T - 1/Y_c, \]
   \[ F_3 = 1/Z_T - 1/Z_c, \]
In the total discount method, the stiffnesses of the failed element are dropped to zero, while some residual stiffnesses are maintained in the limited discount method after element damage. In this research program, the degraded properties of the equivalent damaged element were set to be a constant multiple of the material properties before degradation. The constant, which is given a value between 0 and 1, is called the stiffness reduction coefficient (SRC) [37]. Each damage model was defined by selecting the specified failure criterion and specified stiffness reduction coefficients (SRC1, SRC2, and SRC3) as related to 3-D damage modes (fiber failure, matrix failure and delamination, respectively). In the total discount method SRC=0 while the limited discount method utilized 0<SRC<1. Since SRC1, SRC2, SRC3 can vary from 0 to 1 independently of one another, the material degradation models were defined by three independent variables. It was found to be impractical to consider the whole set of these variables which can vary in a cubic space region. A simplified approach was therefore taken in the present study that was very similar to the method reported by Tan [36]. In the present approach, the values of SRC1, SRC2 and SRC3 were initially set to be the same (i.e., SRC=SRC1=SRC2=SRC3), with SRC varying from 0.0 to 1.0 in a parametric study. The effects of the parameter SRC were evaluated by comparing the numerically predicted damage results [14] to the actual experimental results [19]. The SRC value, which provided the best match with the experimental results, was then identified for both stem designs. Once this was determined, further parametric studies were conducted with SRC=SRC (i.e., fixed value) while SRC2 and SRC3 were varied from 0.0 to 1.0.

The damage development predictions were then recompared with the experimental results to determine which set of parameters most closely matched the actual fatigue damage behavior of the femoral components. By these methods, the results of the parametric progressive failure analyses of the FE models of experimental setup were compared to the actual experimental behavior of the composite hip stems, and these comparisons were then used to identify the one set of damage model parameters which most closely simulated the damage development observed in both of the actual test specimen designs. This process thus represents a calibration of the parametric model to the actual structural performance of laminated femoral components in terms of the type, location, and relative severity of the ply level damage which progressively occurred during component fatigue.


From the global level solutions of the femur/implant model, the hot spot was identified using SED as the criterion. After isolating the hot region, a refined 3-D local model was created in the specified local zone wherein the plies were modeled as having homogenous orthotropic material properties. The local domain was discretized using 20-node brick element or 15-node wedge element, where one brick element or wedge element was used per ply in the thickness direction. The boundary conditions of the local model were interpolated from their corresponding global displacement solution using elemental shape functions. The calibrated
parameter set which was established by the methods described in Step 6 was then applied to this present local model to predict damage behavior for the femur/implant model. Thus, by this approach, the damage behavior of the two femoral component designs representing two ply orientation and stacking sequences was assessed for an in situ laminated composite femoral component under loading conditions simulating the toe-off phase of gait.

3. Numerical results

3.1. Global analysis of femur/implant model

From the global solutions of the femur/implant model under the toe-off loading condition, the femur/implant structure was observed to deform in all three spatial directions: axial compression, bending in the A–P and M–L planes, and twisting in the horizontal plane [14]. Fig. 4 shows the SED distributions in the structure as determined by the global analysis of the femur/implant structure. The global model analysis showed that the hot spot in the composite hip implant was in the anterior–medial neck region of the implant. The concentrated stresses exhibited in the anterior–medial area of the neck of the stem were approximately 30 mm distal to the tip of the stem along the neck axis (the coordinate system is shown in Fig. 1). This localized critical region of SED is a direct result of the combination of the in-plane and out-of-plane applied loading and the stem support conditions. These results agree well with the results from similar studies reported in the literature [7,9–13]. The volume of material immediately surrounding the hot spot region was then selected for detailed local analysis and subsequent damage development modeling.

3.2. Experiment system design and global analysis

The location on the implant to which potting material was placed in the experimental test fixture was designed to be at the same level as the osteotomy line of the femoral model (Fig. 3). In accordance with St. Verant’s Principle, FE analysis showed that there was almost no difference in SED and stress distributions in the hot spot area of the implant with the four different investigated stem support conditions: bonded interface and unsupported distal tip, bonded interface and supported distal tip, non-bonded interface and unsupported distal tip, non-bonded interface and supported distal tip [14]. Stem support conditions were selected that were most easily accomplished experimentally and which were most readily maintained during testing [19]. For the fatigue test, the supported distal tip condition was selected in the experiments with a non-bonded stem/potting medium interface. The distal tip of the stem was supported to prevent significant axial y-displacement of the stem down into the fixture during testing which, if not prevented, would shift the critical region away from the area of interest [19]. It was also found to be necessary to incorporate small flat metal plates bordering the anterior and posterior sides of the stem to prevent large torsional rotation of the implant during loading. This effect not only restrained torsional rotation of the stem, but also prevented the foundation material from being overly plastically deformed during testing [19].

A final design was determined from the overall test fixture design parametric studies to provide a SED distribution at the hot spot of the stem which best matched the femur/implant model [14], as shown by the comparison in Fig. 4. The final design of the experimental test fixture is shown in Fig. 3(b) with a femoral component in place.

This test fixture is similar but not identical to the main existing standard for the fatigue testing of hip prosthesis, ISO 7206. In the ISO 7206 standard, the stem is distally potted in a stainless steel cup with acrylic cement in a position simulating 10° of adduction. The ISO 7206 standard specifies that the component should be mounted in resin or cement to have a distance of 80 mm from the center of the prosthetic head. In such a setup, the part of the stem within the mounting resin carries little or no load and most of the upper part of the stem is lightly loaded compared to the region immediately above the mounting resin [4]. Therefore, it is only really this region that is properly tested [4]. Clearly, this may not be appropriate for testing composite implants. In our design, we modified the potting regions and the loading directions compared to this ISO standard. The composite prosthesis was supported and loaded by a parametrically designed fixture to attain the stress distributions in the implant that most closely matched the simulated in vivo implanted femur model. Tests with
this modified fixture should therefore be more realistic than that with the ISO 7206 system.

3.3. Global/local analysis of FE model of experimental system

As described in the previous section, the FE model of the experimental system (fixture/stem structure) showed that the hot spot in the stem occurs in the anterior-medial aspect of the neck for the toe-off loading case, which is the same result obtained in the global analysis of femur/implant model. This is also similar to positions where failure has been observed in laminated composite femoral components reported in the literature, including the reported clinical failure [7]. Once identified, the hot spot area from the global model was isolated as shown in Fig. 5. Fig. 5 presents an illustration of the development of the local FE model of this hot spot region and the surrounding portion of the potting medium or femur.

From the numerical results of the stress distributions in the Design I and Design II stems with the local detailed analysis, all stress components showed large ply to ply variations in magnitude due to the variations in ply orientation [14], as expected. This underscores the point that local ply level stress analysis is necessary before accurate estimates of comparative strengths may be made for laminated composite femoral components [9–17].

3.4. Experimental testing of laminated composite femoral components

In the experimental program, the fatigue load levels were specified to be 90%, 80% and 70% of the average static damage initiation load based on static tests on the samples of the two different designs with the ply orientations shown in Step 3 [19]. Experimental results showed that the damage initiation and propagation experienced in the two laminate designs are consistent among all the load levels [19]. Examples of the experimental fatigue results are shown in Fig. 6. Detailed results of the fatigue test are reported in Refs. [19,21]. The damage in the stem Design I was found to initiate at an earlier stage, but to propagate more slowly across the primary load bearing plies as compared to Design II. The damage in Design I was concentrated mainly in the anterior-medial neck region of the stem (Fig. 6(a) and (b)). Matrix cracking was followed by fiber microbuckling at an early stage of fatigue loading cycles, but then exhibited very slow/restrained growth throughout the life of the specimen. There was little growth of this damage as cyclic loading continued, and only very localized delamination on the anterior–medial surface developed by the end of the fatigue test. In Design II, damage mainly occurred in the form of matrix cracking and fiber microbuckling early in the test (Fig. 6(c) and (d)). Unlike damage in Design I, the damage in Design II propagated rapidly following the failure of the primary load bearing plies. Although damage was concentrated initially on the anterior neck surface, it quickly grew transversely across adjacent plies and towards the medial side of the laminate. Matrix cracking was followed almost immediately by fiber microbuckling, secondary cracking, and localized delaminations. Microbuckling occurred initially in the primary load bearing plies across the entire anterior surface at the boundary interface of the stem/potting material. Damage growth was rapid and significant at an early stage in the fatigue test. It is evident that Design I exhibited greater fatigue resistance and outperformed Design II in the experimental fatigue tests for this
research with Design I exhibitory damage propagation controlled fatigue life while Design II was damage initiation controlled [19,21].

3.5. Progressive failure analysis of FE model of experimental system

As determined by the global models of the stem in both the femur and experimental test fixture, stress concentrations were found to occur in the stems slightly above the osteotomy and fixture potting level line, which agreed well with experimental observations [19]; again, this indicated the appropriateness of the global analysis. From the local models, damage was predicted to be initiated in the stem surfaces in this high-stress region and to propagate inside the stem and along the plies [14].

The predictions of the ply level damage states for each of the various sets of failure and damage model parameters were compared to the fatigue test results to calibrate the numerical damage model. From the numerical parametric analysis results, the predictions using the independent failure criterion with stiffness reduction coefficients SRC1 = 0.01, SRC2 = 0.01 and SRC3 = 0.01 were determined to best match the experimental results [14]. This parameter set was therefore selected as the one best calibrated to the actual damage behavior of the composite femoral components for the final portion of this overall program, which was the application of the failure model for the progressive failure analysis of the anatomic femur/implant model.

From the local model numerical analysis, initial damage was predicted to occur at the interfacial boundary of the stem/potting material in these two designs. The damage was also predicted to concentrate on the anterior–medial side of the stem, the same location that was observed experimentally [19]. The different stacking sequences for Stem Designs I and II provided distinctly different damage development behaviors (Fig. 7), as expected, due to their different ply level parameters. For Design I, initial fiber damage was predicted in the +30° ply (per the global coordinate system as shown in Fig. 1) on the outside of the anterior neck surface. This corresponds to −1° with respect to a local coordinate system with 0° defined by a vector parallel to the free edge of the ply pointing toward the femoral head. There also existed fiber damage in the next +60° plies moving toward the center of the laminate. There were more damage locations throughout this design because of the high number of +60° plies. Unlike damage in Design I, damage in Design II was predicted to propagate from the +60° plies to other adjacent +0° and ±45° plies. Damage was evident across the entire thickness of the laminate in the local model, from the anterior surface to the center of the laminate in Design II.

In comparison to Design II, it is evident that Design I initiated damage at a slightly lower load level than Design II, but Design I demonstrated its ability to restrain damage growth more than Design II, while damage in Design II exhibited more of a catastrophic effect during damage propagation. It is evident from these results that Design I should be more fatigue resistant than Design II. This result agreed very well with the experimental observations [19].

3.6. Progressive failure analysis of in situ composite femoral components

As previously described, the hot spot in the femur/implant model was identified to occur in the anterior–medial aspect of the neck of the femoral component for the toe-off loading case. Again, this is similar to positions where failure has been previously observed as reported in the literature. This high-stress area was thus isolated by a ply level local model, as shown in Fig. 5.

As discussed previously for the progressive failure analysis of FE model of the experimental system,

![Fig. 7. Damage accumulations in the composite stems of the fixture/stem models (SDV1, SDV2, and SDV3 represent the failure indexes for fiber failure, matrix failure, and interlaminar delamination failure, respectively).](image-url)
parametric studies showed that the independent failure criterion and SRC set of SRC1 = 0.01, SRC2 = 0.01, SRC3 = 0.01 predicted damage behavior which best matched the actual experimental fatigue results for each stem design. Therefore, this parameter set was applied for the progressive failure analysis of the femur/implant model. Fig. 8 shows the damage development of the implant in the two different designs (Designs I and II) of different ply orientations and stacking sequences in the femur/implant model. Ply damage in Design II was shown to be more concentrated and continuous over the critical zone cross section of the stem, thus unloading the volume of material above and below the damage area. The more widely dispersed damage in Damage I is indicative of the ability of the damaged plies to continue to transfer load across the critical zone cross section of the stem, thus showing a greater degree of structural integrity. Comparison of these results with the results from the progressive failure analysis of the FE model of the experimental setup indicates very similar damage initiation and propagation behavior, as expected, although the foundation conditions are very different [14,19]. However, damage initiation and propagation occurred differently between the two different stem designs. This indicates that the ply orientation and stacking sequence have very important effects on the structural damage behavior, as anticipated. Again, both these structures were predicted to fail in a fiber compression damage mode. As in the experimental system, Design I was shown to be able to better restrain damage growth compared to Design II, which exhibited a catastrophic level of damage propagation. Therefore, from these results, Design I is predicted to be more fatigue resistant than Design II under the applied loading conditions.

4. Discussion and conclusion

From the global level solutions of the FE models of both the femur/implant structure and the experimental system, the critical region limiting fatigue life was identified, using SED, to occur in the neck region of the implant. This is similar to positions where failure was observed in experimental fatigue tests conducted in parallel to this research, and reported to have occurred during clinical trials with laminated composite femoral components. It is encouraging to note that a scalar criterion such as the SED has the capability to correctly predict sites where failure occurs. This shows the usefulness of global level analysis for implant design with these materials. However, the numerical results from the ply level local analyses have shown striking differences in the stress distributions in Designs I and II. These two designs have the same global stiffness properties, but have different ply orientations and stacking sequences. Comparing damage developments in these two designs within the modeled experimental system, these two designs were also predicted to have very different damage development characteristics. Therefore, the fatigue strength of the laminated composite femoral component is shown to be influenced by laminate ply orientation and stacking sequence, as anticipated. This implies that ply level stress analysis is necessary for composite femoral component fatigue strength analysis and design optimization.

Although there are a number of methods available that allow the contextual analysis of structural designs made from composite materials, the complex and fully 3-D nature of the structural response while using composites in THA applications makes straightforward 3-D stress analysis computationally intractable while simplified 2-D analysis is not sufficiently accurate. A 3-D global/local method was therefore developed to overcome this difficulty. From the numerical results of the femur/implant structure, this developed 3-D global/local method was shown to be suitable for locating the critical region within the composite femoral component, providing 3-D ply level stress distributions, and predicting ply level damage initiation and propagation behavior efficiently and accurately.

As an integral part of this program, an experimental fixture for fatigue testing was designed to simulate the loading conditions experienced by an in situ composite femoral component. By adequately adjusting test parameters of the experimental system, especially the stem head load vector direction, the SED distributions between the experimental system and femur/implant structure could be matched very closely. This global critical response match provides the basis for comparing the structural responses of two stem designs. In order to provide experimental verification data for the numerical predictions, fatigue experiments were performed to
monitor and document damage initiation and propagation in the two different stem designs in the experimental fixture. These experimental results were then used for damage model calibration and for comparison of the two implant designs.

It has generally been accepted that damage models using different failure criteria and different material degradation model parameters will predict different damage development behavior. The effects of these parameters were demonstrated in the FE failure analysis of the experimental fatigue test fixture. Through parametric studies of damage development with different damage models, excellent agreement was obtained between predicted and actual damage development within the composite femoral component using the independent failure criterion with stiffness degradation coefficients of \( SRC1 = 0.01 \), \( SRC2 = 0.01 \) and \( SRC3 = 0.01 \). From the numerical results, similar damage initiation and propagation behavior were predicted to occur in each of these two designs between the stem in the bony foundation of the femur/implant model and in the polymer resin plastic foundation of the experimental model. This result again implies that the internal design characteristics (i.e., ply orientation and stacking sequence) have very important effects on the structural damage behavior in thick laminated composite structures. This also indicates that laminated composite femoral components should be able to be designed with improved damage resistance by optimizing the laminate ply orientation and stacking sequence to maximize fatigue damage resistance.

In order to design a composite hip prosthesis adequately, the following important issues must be considered: (1) stress shielding; (2) stem/bone interface stresses; and (3) prosthesis strength. In hip replacements, a compliant stem design may decrease stress-shielding effects, however, it may induce other problems, such as increased stem–bone interface shear stresses. The simple substitution of a lower modulus material may not be appropriate in hip replacements [4]. It is therefore necessary to fully understand the theoretical basics for the design of the composite component and develop analysis and design approaches to consider these conflicting objectives and limitations to achieve a reasonable design. Prior to this paper, a multi-level optimization methodology to design heterogeneous laminated composite hip prostheses for total hip arthroplasty was developed [6,17]. In the first level of the optimization methodology, the objective was to minimize the stress shielding in the calcar region by designing global stiffness. The maximum stem/bone interfacial shear stresses were constrained to be below a user-defined limit of the interface strength. The calcar region was selected because it is the area that is most likely to experience stress-shielding induced bone resorption following hip arthroplasty. The objective in second level optimization was to minimize the maximum shear stresses along the femur/stem interface while enforcing the same level of stress states in the calcar by changing laminate orientation and stacking sequence. The method developed in this present paper extends this previously developed optimization methodology, which can be considered as a third level in the optimization procedure. The final objective is to maximize the composite fatigue strength of the critical region with the progressive failure analysis method, while minimizing the stress shielding effects and keeping the interfacial stresses below user-defined maximum levels.

In conclusion, the present research provides an FE method to accurately and efficiently predict the damage response of a laminated composite implant as a function of the ply level design parameters. This research can lead to the use of the analytical models as design optimization tools for composite femoral components prior to conducting extensive experimental tests. Furthermore, the above method is not specifically linked to femoral component analysis but should have broad application for the design of other complex composite structures for both medical and non-medical structural applications.

Acknowledgements

This research was supported by the Department of Veterans Affairs, Veterans Health Administration, Rehabilitation Research and Development Service, Ralph H. Johnson Medical Center, Charleston, SC. The authors would like to thank Dr. Sundar Srinivasan, Dr. Wei Fu, Dr. Dan Warner and Dr. Tom Eason for their helpful discussions and suggestions throughout the course of this study.

References


