ELSEVIER

International Journal of Fatigue 24 (2002) 739–746

International Journal of Fatigue

www.elsevier.com/locate/ijfatigue

Fatigue analysis of cemented hip prosthesis: damage accumulation scenario and sensitivity analysis

P. Colombi *

Department of Structural Engineering, Milan Technical University, Piazza L. da Vinci, 32, 20133 Milan, Italy

Received 20 November 2000; received in revised form 14 September 2001; accepted 19 October 2001

Abstract

This work is dedicated to the fatigue analysis of cemented total hip arthroplasty. In particular the damage evaluation scenario is simulated and a sensitivity analysis is performed. To this end, two different damage evaluation algorithms (the elasto-brittle and the continuous damage one) are proposed and implemented in the finite element code ABAQUS®. Some global damage criteria are introduced to quantify the damage accumulation. The continuous damage algorithm is shown to perform better compared to the elasto-brittle damage one in the estimation of the fatigue lifetime of the cement mantle. A sensitivity analysis is then performed as a function of the cement Young's modulus, the stem–cement friction coefficient and the stem Young's modulus. Numerical results show a significant sensitivity to variations of the cement Young's modulus and stem–cement friction coefficient and a moderate sensitivity to the stem Young's modulus.

Keywords: Fatigue damage; Damage-induced anisotropy; Hip prosthesis; Bone cement

1. Introduction

Forces applied to the prosthesis due to human activity generate complex multiaxial stresses varying in time and resulting in the accumulation of mechanical damage in materials and interfaces [17]. In the literature, finite element models in association with cement damage evaluation algorithms are used to simulate fatigue damage accumulation of a cemented total hip arthroplasty [16,6]. In the same way in this work a simulation strategy is described and implemented in the finite element code ABAQUS® [2]. The goal is the simulation of the damage evaluation scenario in the cement mantle together with a fatigue sensitivity analysis [15]. Two different damage rules are introduced in this paper. The first one is the well known Miner rule (elasto-brittle damage evaluation algorithm), originally presented in Verdonschot et al. [16], which does not take into account the damage-induced anisotropy on the cement stiffness. In Verdonschot et al. [16], this produced a conservative estimation of the fatigue lifetime. Some relaxing assumptions were introduced in Dolinski [6], in order to include the effect of the damage-induced anisotropy and a non-linear damage rule was proposed (continuous damage evaluation algorithm). In this paper such an algorithm is fitted to the quasi-three-dimensional finite element model of the implant [4]. Two different global damage criteria are considered in order to study the fatigue behaviour of the cement mantle. The first one is the mean damage accumulated in the cement mantle while the second one is the stem subsidence in the cement mantle. A validation of the quasi-three-dimensional finite element model of the implant is first pursued in term of peak stress in the cement mantle. The proposed global damage criteria are then used to quantify and to analyse the damage initiation and accumulation phenomenon. A sensitivity analysis is finally performed in order to investigate the influence of some parameters such as the cement Young's modulus, the stem–cement friction coefficient and the stem Young's modulus on the fatigue damage scenario.

* Corresponding author. Tel.: +39-02-2399-4280; fax: +39-02-2399-4369.

E-mail address: colombi@stru.polimi.it (P. Colombi).

0142-1123/02/$ - see front matter
PII: S0142-1123(01)00204-3

2. Finite element model and damage evaluation algorithms

2.1. Finite element model

In order to reduce the high computational costs [12] connected to a fully-three-dimensional model, a quasi-three-dimensional model is introduced. It is able to capture the relevant futures of the implant fatigue damage evaluation. The model is reported in Fig. 1 together with its cross section. Bone, cement and steel prosthesis are appropriately modelled by plane stress elements and interface elements are introduced to simulate the debonded stem–cement interface. The side-plate concept [14,18] is used to account for the three-dimensional structural integrity of the cement and cortical bone. Moreover, the hope stress under the frictional stem–cement interface is recovered by the model as membrane stresses σ_x in the corresponding side-plates that are superimposed on the finite element mesh (Fig. 1). The thickness of the cortical bone and bone cement elements (Fig. 1) are calibrated in order to achieve bending stiffness and membrane stresses σ_x equal to the three-dimensional one. The model contains 2384 4-node plane stress elements and 2824 nodal points.

2.2. Damage evaluation algorithms

In the finite element model the materials (bone, metal and cement) are considered to be elastic and initially isotropic. In the following the proposed damage evaluation algorithms are summarised. Two different damage rules, a linear and a non-linear one, are used to evaluate the fatigue damage evaluation in the cement mantle. The linear damage rule corresponds to the well known Miner rule [16]:

$$D=D_0+n\cdot\frac{\Delta s^m}{c} \tag{1}$$

where D is the damage, D_0 the accumulated damage, n the number of cycles, Δs the stress range and c, m are the material parameters of the underlying S–N curve. In the non-linear damage accumulation rule it is postulated [6] that the damage rate $\dot{D}$ is given by:

$$\dot{D}=c_0\cdot\left(\frac{\Delta s}{1-D}\right)^{m_0} \tag{2}$$

where $m_0=m$ and $c_0=c\cdot(m+1)$ and again Δs is the stress range. By integrating Eq. (2) with initial condition $D=D_0$ when $n=0$, the following relationship is obtained:

$$D=1-\sqrt[m_0+1]{(1-D_0)^{m_0+1}-\Delta s^{m_0}\cdot C_0\cdot(m_0+1)\cdot n} \tag{3}$$

Assuming the principal directions of the stress tensor to be unchanged during the following load cycles [6], the range of the principal stresses, $\Delta s^{ip}=[\Delta s_{I}^{ip},\Delta s_{II}^{ip}]$, in every integration point, ip, is evaluated. The updated damage tensor, D_{local}^{up}, in the local principal stresses reference system, at each integration point, is then computed

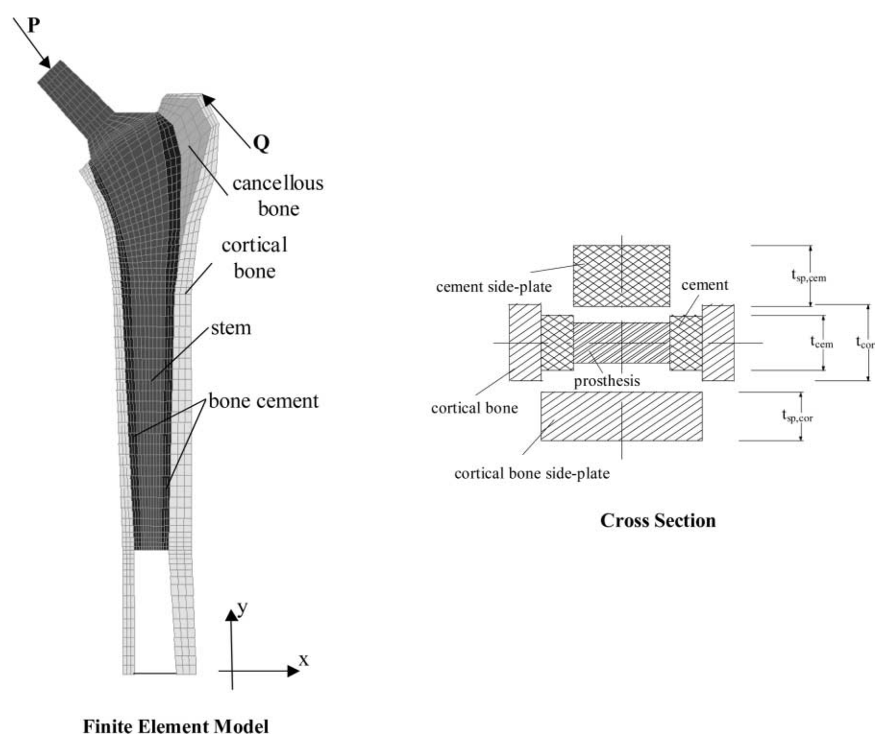

Fig. 1. Finite element mesh and section of the quasi-three-dimensional model of the implant.