Finite Element Analysis on Vibration Modes of Femur Bone

Arpan Gupta¹, Kwong Ming Tse²

¹ Department of Mechanical Engineering, Graphic Era University, Dehradun, India
² Department of Mechanical Engineering, National University of Singapore, Singapore

Email: ¹apn.gpt@gmail.com, ²tsekm@nus.edu.sg, ²tsekm.research@yahoo.com

Abstract— Femur bone is the longest and largest bone found in human body. It is also one of the most commonly fractured bones of human body, especially for the elderly. In this paper, vibration analysis of femur bone is studied by the help of finite element simulation to provide more insight in designing bio-aided equipments or protective sports equipments for femur. The simulations were performed using the software – Abaqus. The vibration patterns for first twenty modes were studied. The mode shapes show that the natural frequency of vibration varies from 722 Hz to 8480 Hz. The results were compared with experimental results presented in literature. External excitation on the femur bone must be avoided to coincide with these natural frequencies; otherwise it could lead to fracture of bone.

Index Terms— finite element method, femur bone, biomechanics, vibration modes, natural frequency

I. INTRODUCTION

Biomechanics involves applying principles from mechanical engineering to biological systems. It may involve evaluation of stresses, strains, vibration patterns as well as analyses of fluid flow, kinetics and dynamics of mechanisms present in human body. There are many tools to perform such analyses, out of which finite element analysis (FEA) is very popular tool for performing static and dynamic loading of a human organ such as bone structure.

Finite element analysis [1] is a computer based method for performing numerical analysis which can be used to analyze structures of complicated geometry and inhomogeneous material properties. Finite element method is widely accepted and used as an alternative tool for biomechanics modeling [2] which has complicated geometrical shapes and heterogeneous material properties.

One of the authors (Tse, KM) has shown various applications of FE simulation [3]--[5] and computational fluid dynamics [6], [7] in various fields of biomechanics. Author has also performed a vibration analysis of human head-brain-neck model [8].

The femur bone is the longest and the largest bone found in the human body. It is also known as the thigh bone. It connects to the pelvis at the proximal end to form the hip joint and to the tibia at the distal end to form the knee joint. Femur bone of human body, which takes the largest percentage of the body weight, is one of the most commonly fractured bone in human body. Hence, extra care and special consideration are
required when designing bio-aided equipments or sport protective equipments to avoid the coincidence of the resonant frequencies and the external excitation frequencies.

For the past few decades, tremendous effort had been spent on determining the natural frequencies of femur bone. Earlier research on femur bone vibration was mainly based on experimental tests involving animals and cadavers. FEA offers a cost-effective alternative in the biomechanical studies of vibration analysis of femur. Researchers have been studying the vibration characteristic of femur bone since 1980s. Khalil et al. [9] obtained natural frequencies and mode shapes of femur bone using experimental and analytical methods. The experimental measurements were made based upon the Fourier analysis of transfer function. For analytical solution, a mathematical model consisting of 59 elements was analyzed using transfer matrix method. The model that they considered was a freely vibrating bone, which is not the real case. In a real structure, bone is constrained between pelvis and tibia. Therefore such a boundary condition is not justifiable. The first twenty experimental natural frequency of femur bone for free-free boundary condition was found to vary from 250 Hz to 7300 Hz. The analytical solution to natural frequency also lies in the same range. However, for the longitudinal mode of vibration, the natural frequencies were 2118 Hz, 4407 Hz and 7264 Hz. In this work, our focus will be on the longitudinal mode of vibration using FEA. Similar to Khalil et al. [9], Hight et al. [10] performed vibration analysis of human tibia using beam type finite element model and compared with analytical solution and experiment. Dynamic loading of femur bone with stress wave was studied by Richard and Subrata [11]. They used the loading of impact by a steel ball at one end and measuring the measuring the stress wave propagation by two strain gauges. Thomas et al. [12] has studied the effect of mechanical vibration on human femur. However, they considered a low frequency range from 0 Hz – 500 Hz. Researchers have also used ultrasonic techniques for measuring elastic properties of human bone [13]. Impact response on bone has also been predicted by using simplified finite element model [14]. Finite element simulation were also used to determine orthotropic material properties using modal analysis [15]. Fracture analysis of a femur bone has also been predicted by finite element simulation [16]–[18].

In this present study, modal responses of the femur bone, in terms of eigenfrequencies and mode shapes, are determined using modal analysis.

II. MATERIAL AND METHOD

Finite element (FE) simulation prediction depends entirely upon the geometry considered, the loading conditions and boundary conditions. For this reason, accurate FE model of femur bone with accurate geometry are very important. The geometrical information of a patient-specific femur bone was obtained from CT data by Nuaalsy (Nanjing University of Aeronautics and Astronautics, China), using Mimics (Materialise, Leuven, Belgium). A FE model based on this geometrical information was then reconstructed with 50601 linear tetrahedral elements of average element size of 3.2 mm and average aspect ratio of 1.56, using an advancing-front meshing algorithm – Hypermesh v11.0 (Altair HyperWorks, Troy, MI, USA) (Figure 1).

![Figure 1: (A) Anatomy and (B) Meshes of the human left femur bone model.](image)

The femur bone was assumed to have isotropic linear elastic material properties. The density was chosen to be 866 kg/m³ based on the samples’ average of the largest elderly group [19] while the Poisson’s ratio was taken as 0.4 [20]. The Young’s modulus was computed from the density using the reported correlation in [21] and was found to be 7.585 GPa. To replicate the restriction of motion by the adjacent bones, fixed boundary conditions were applied at the base of both the lateral condyle and the median condyle as well as around the femur neck.
In order to determining modal responses, modal analysis using FE is performed using implicit FE code – Abaqus v6.10 (Dassault Systèmes Technologies, RI, US). The governing equation of the dynamic response is given as follows:

\[
[M][\ddot{x}] + [C][\dot{x}] + [K][x] = 0
\]  

(1)

where \([M]\), \([C]\) and \([K]\) are the global mass, damping and stiffness matrices of the model; \(\{\ddot{x}\}, \{\dot{x}\}\) and \(\{x\}\) are the nodal acceleration, velocity and displacement vectors respectively.

For undamped free vibration (i.e. \([C] = 0\)), the solution of the above equation can be written as follows.

\[
\{x\} = \{X\}e^{i\omega t}
\]  

(2)

where \(\{X\}\) represents the amplitudes of all the masses (mode shapes or eigenvectors) and \(\omega = 2\pi f\) represents each eigenvector’s corresponding eigenfrequency in rads\(^{-1}\), while \(f\) represents the natural frequency in hertz. Thus the governing equation mentioned above reduces to:

\[
[K] - \omega^2[M] = 0
\]  

(3)

The above equation is known as eigenvalue problem in matrix algebra and is considered as linear by replacing \(\omega^2\) by \(\lambda\). The system solution, which relies on determining each eigenvector, with its corresponding eigenvalues solved by the natural frequency extraction using Lanczos eigensolver in Abaqus.

III. RESULTS

This section presents result on the dynamic characteristics (resonant frequencies and mode shapes) of the 3D FE model of femur bone. By performing modal analysis on this FE femur model using the Abaqus implicit code, twenty resonant frequencies and associated mode shapes are recognized in the frequency band of 700 to 8500 Hz (Table 1 & Figure 2).

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Natural Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>722.66</td>
</tr>
<tr>
<td>2</td>
<td>800.50</td>
</tr>
<tr>
<td>3</td>
<td>1584.18</td>
</tr>
<tr>
<td>4</td>
<td>1771.43</td>
</tr>
<tr>
<td>5</td>
<td>2672.4</td>
</tr>
<tr>
<td>6</td>
<td>2857.11</td>
</tr>
<tr>
<td>7</td>
<td>3513.23</td>
</tr>
<tr>
<td>8</td>
<td>3988.55</td>
</tr>
<tr>
<td>9</td>
<td>4078.11</td>
</tr>
<tr>
<td>10</td>
<td>4695.44</td>
</tr>
<tr>
<td>11</td>
<td>4890.35</td>
</tr>
<tr>
<td>12</td>
<td>5798.52</td>
</tr>
<tr>
<td>13</td>
<td>5986.93</td>
</tr>
<tr>
<td>14</td>
<td>6365.08</td>
</tr>
<tr>
<td>15</td>
<td>6439.11</td>
</tr>
<tr>
<td>16</td>
<td>7129.54</td>
</tr>
<tr>
<td>17</td>
<td>7249.07</td>
</tr>
<tr>
<td>18</td>
<td>7474.80</td>
</tr>
<tr>
<td>19</td>
<td>8141.32</td>
</tr>
<tr>
<td>20</td>
<td>8479.95</td>
</tr>
</tbody>
</table>
These computed resonant frequencies of the femur bone in the undamped free vibration are tabulated in Table 1, with its first resonant frequency at 722.66 Hz. Within the twenty mode shapes, nine various, distinguishable mode shapes of the femur model are recognized, namely the first five sinusoidal lateral buckling modes (Mode 1, Mode 3, Mode 5, Mode 12 and Mode 18) as well as some constitute of localized expanded/elongated modes (Mode 7, Mode 14, Mode 15 and Mode 19) (Figure 2). The sinusoidal lateral buckling mode shapes of the femur model dominate throughout its first twenty modes and are identical to some subsequent mode shapes, except for the difference in the magnitude of relative displacement.

IV. DISCUSSIONS

External loading or excitation on the human body can have severe consequences if the external excitation frequency matches with the natural frequency of the bone. In this work, it is found the natural frequency for first twenty modes of femur bone varies from 722 Hz to 8480 Hz. The results are also in good agreement with experimental work by Khalil et al. [9]. In his work, the longitudinal modes of vibration were identified to have natural frequencies as 2118 Hz, 4407 Hz and 7264 Hz which are close to 5th, 10th and 17th mode predicted by FE simulation. The boundary condition in the experiment was free-free condition, which is not the real case. And therefore the deviation in the results can be thus explained.

In any impact loading or excitation, it should be so designed that the excitation frequency does not coincides with the natural frequency of the femur bone. Otherwise, it could lead to fracture of the bone. The fracture location at a particular frequency can also be predicted from the mode shape in Figure 2. For e.g., at 722.6 Hz, the bone is likely to fracture from the middle (region of maximum displacement and therefore stress). Similarly external excitation at 6439 Hz (at Mode 15) is likely to fracture the bone from the femur head.

V. CONCLUSION

In this work, FEA of femur bone was performed using Abaqus. The natural frequencies and mode shapes for femur bone were identified for fixed-fixed boundary condition. The results were compared with the experimental results in literature and they were in good agreement. While designing any biomechanical equipment, or sports equipment, care should be taken that external excitation does not coincide with the natural frequency of the femur bone as predicted. Else, the excitation can lead to fracture of the bone which can be predicted by the mode shape at the corresponding natural frequency.

REFERENCES


