Proceedings of the ASME 2012 International Mechanical Engineering Congress & Exposition
IMECE2012
November 9-15, 2012, Houston, Texas, USA

IMECE2012-89192

FEA Analysis of a Portable Knee Rehabilitation Device Using Mechanical Loading

Daric Fitzwater, Andrew Rophie,
Benjamin Schroeder,
Andrew Dole, Juan Solano,
Joshua Keith
Indiana University Purdue University Indianapolis
Indianapolis, IN, USA

Sohel Anwar
Indiana University Purdue University Indianapolis
Indianapolis, IN, USA
soanwar@iupui.edu

Hiroki Yokota
Indiana University Purdue University Indianapolis
Indianapolis, IN, USA
hyokota@iupui.edu

Stanley Chien
Indiana University Purdue University Indianapolis
Indianapolis, IN, USA
schien@iupui.edu

ABSTRACT

In this paper, a solid model has been created with CAD software and analyzed with FEA software to obtain the deformed geometry, stress distribution, modal frequencies, temperature distribution, and life expectancy of a knee loading device that will be used in a combined biomedical and mechanical engineering research initiative. The purpose of this device is to mechanically load the end of the long bone of the human leg, causing movement of the fluids within the bone that can stimulate increased growth of bone tissues. This could potentially be used to speed the healing process of bone fractures.The CAD model of the device was constructed in Pro/ENGINEER and then exported to ANSYS Workbench where it was then meshed and solved using the finite element method.

INTRODUCTION

The knee loading device examined in this paper only remains effective for small levels of deformation. The intended displacement of the working device is very small, a maximum of only 6.35mm, and therefore the device cannot accommodate large deformations as such deformations will greatly decrease the effective range of motion of the device. Furthermore, the device will be used in testing situations where precise force and frequency applications are essential. For example, damping caused by deformation of the materials of the device will alter measurements and lead to inaccurate calibration and application of the device.

The device will undergo noticeable stresses, particularly near the pivot points of the mechanism. Should the stresses be too great for the material selected, the device may yield, resulting in a mechanical failure. Since the device will be providing a cyclic loading, the device will be subjected to the low operation frequencies used. Should the frequency of operation coincide with one of the modal frequencies, the device will fail.

The voice coil linear actuator used to power this device gradually heats up overtime. Some concerns have arisen from this observation concerning user comfort and safety. The user must be kept sufficiently insulated from the motor so that he or she is unaffected by the temperature rise even in the worst case scenario.

Since it is currently planned to patent and potentially commercialize the device, the device must have a sufficiently long lifespan to be considered durable. The device would potentially be used for several minutes on a daily basis for prolonged periods and should be able to last several years without failing due to fatigue.

ANSYS Workbench 13.0 was used to analyze the deformed geometry, stress distribution, modal frequencies, temperature distribution, and life expectancy of the knee loading device for the reasons discussed above. A model from Pro/ENGINEER 4.0 was imported into ANSYS for this analysis. Design recommendations are made based on the findings.

DEVICE BACKGROUND

The healing process of long bones can be stimulated by mechanical means. Mechanical deformation of the soft tissue at the end of the long bones creates an internal gradient of hydraulic pressure within the bone, thus causing the movement of fluid throughout the porosity of the bone. This fluid shear stimulates osteocytes (bone cells), in a manner dependent on the magnitude of the shear stress. The potentially-enhanced osteogenesis caused by such tissue stimulation can decrease the healing time of fractures within the bone as well as increase the density of the bone, something that could prove useful in the prevention or treatment of osteoporosis. Several areas of research have opened within the field of biomedical engineering relating to the purposeful stimulation of bone growth. Figure 1 depicts the principles of mechanical loading and fluid movement within a bone.

The role of neural signaling and blood circulation in load-driven bone formation is an important subject but not well understood. There are other loading modalities such as whole body vibration, which employ mechanical loading in a form of low-magnitude, high-frequency vibration. The described joint loading modality is, however, an unconventional method, which applies lateral loads to synovial joints. The mechanism for enhancing bone formation with joint loading is proposed to be different from that with whole body vibration as depicted in Fig. 1 and described in the review article reference [1]. Little is known about its effects on the neural or circulatory systems.

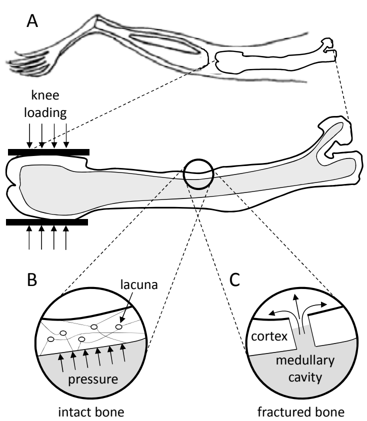

Fig. 1.Depiction of fluid flow, pressure, and deformation within a mouse bone [1].

Several devices have been commercially developed that have been marketed as bone stimulating devices for healing purposes. These devices operate in a number of ways. Several companies produce ultrasonic vibrators, like the Exogen 2000+, that wrap around a limb and vibrate the fluid within [2]. Other devices, such as the Physio-Stim Lite system, use electromagnetic pulses to induce electric fields within the bone and thereby stimulate the tissue by inducing a low-magnitude, high-frequency load [3]. Such devices do see use and are approved for treatment of orthopedic and neurosurgical conditions [4, 5]. However, a device does not


Fig. 2.Four membership functions used in fuzzy system.

exist that applies cyclical mechanical loading at larger magnitudes for use on humans.

Depending on loading conditions (durations, frequencies, and amplitudes), vibrations is potentially dangerous since hand-arm vibration syndrome (HAVS) can be induced if applied to an upperlimb [15]. It is important to evaluate loading conditions that do not present any risk of low blood circulation or nerve damage.

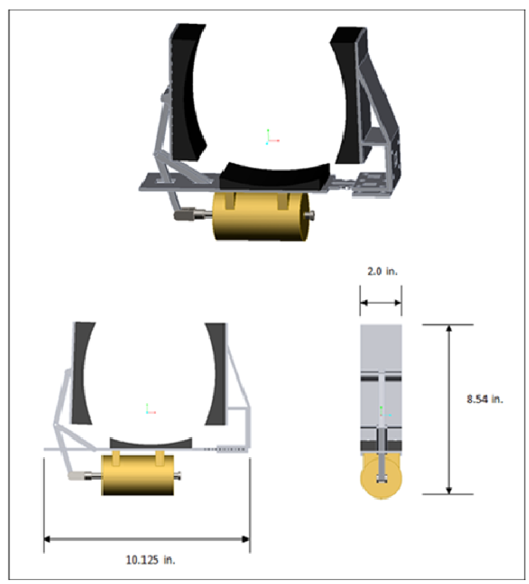

Fig.3. Geometry and general dimensions.

To apply such a load, a device would need to have a means of producing a transverse force directly to the end of a long bone, such as at the knee. A cyclic force applied in such an area would force a slight shift of the fluid within the bone towards the oposite end of the bone in a controlled fashion. While no such device currently exists for use on humans, a new joint studyseeks to develop a portable device designed for human use to be used in future testing. While such a device has not been used for humans, research has been performed in the past which involved the use of similar devices on animals while trying to determine the degree of effectiveness of such treatment [6-8]. These experiments provided positive results.

Upon the success of past devices used in animal studies, a second device was produced as a part of a multidisciplinary engineering research project. This device was used as a first-generation preliminary concept that used an electric brush motor, worm gear, and cam to provide a cyclic loading device that could feasibly be used on a human knee. While this device provided a point of reference, it was not easily transported and lacked a designated microcontroller.

A third-generation device is currently being designed and prototyped that will incorporate a mechanism in addition to a voice coil linear actuator. The mechanism will allow the motor to be placedbelow the device where it will not create an unbalance. In the model of this new device, shown in Figure 2, only the frame and functional components are presented, without the thin plastic covering and knee strap that would be present,[for aesthetic and safety concerns] were the device in actual operation. The model of this device will be shown and discussed in greater detail subsequently.

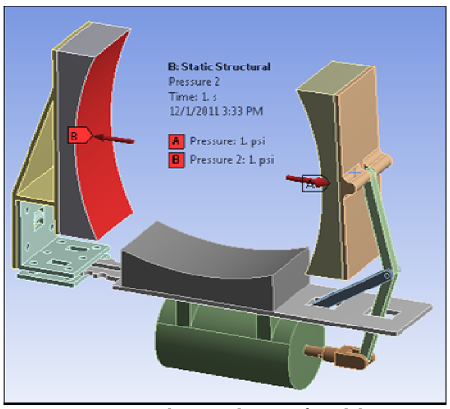

Fig. 4.Loading conditions of model.

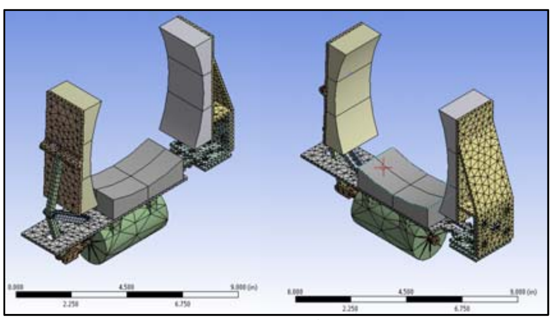
Fig. 5.Compartmentalized meshing for static loading analysis.

MODEL AND ANALYSIS SETUP

The CAD package used to model the knee loading device was Pro/ENGINEER. Figure 3 above shows the final assembly with the main dimensions to demonstrate the overall special footprint of the device. Once the model was assembled in Pro/Engineer, it was imported into ANSYS and the next step was to apply the material properties of the device. Steel was used for most of the device. The only parts that were not steel were the three contact pads (shown in black in Figure 3) which were instead polyethylene.The properties for the steel used in the device were for structural steel. ANSYS has sets of built-in properties for most common materials including the general-purpose low-carbon steel and high-density poly-ethylene used for this analysis [9].

The next step after setting the material properties is to apply the loading conditions to the model. Figure 4 depicts the loading conditions used for the static load analysis. As can be seen from the figure a pressure of approximately 6.90KPawas used as the load for each vertical pad. The pressure equates to approximately 40 N over the entire surface area which is the desired maximum load for the device. The base plate of the device was used as the fixed support of the system.

The coefficient of convection was chosen for ambient temperature and natural convection. A casing is planned for the implemented design. However, ample openings are planned for the area of the casing surrounding the motor to provide ventilation so that the temperature of the air surrounding the motor does not significantly rise, making this a fair approximation.The values for convection coefficient was chosen as 0.05258 W/(m^2*K).

The last step for the model was to develop the discretization mesh. In order to minimize the processing time, the mesh was compartmentalized so that only the critical components in the analysis had a fine mesh and the other components had a coarser mesh. Figure 5 above shows how the model was meshed for the static analysis model. A tetrahedral mesh was used for this analysis.

A. *Finite Element Stress Analysis*

The main relationship behind all finite stress analysis, whether it is one-dimensional, two-dimensional or three-dimensional, is:

$$\{F\} = [K]\{U\} \tag{1}$$

where {F} corresponds to the force vector at each node, [K] represents the stiffness matrix of all elements, and {U} represents the displacement vector of each node in a structure [10]. Equation 1, comes from the basic principle of stress and strain were stress is equal to force divided by cross-sectional area and strain is equal to change in length divided by original length. Stress and strain are related by Hooke's Law, which states that stress is directly related to strain by a factor known as the Modulus of Elasticity, which is unique to every material.

The principle behind finite element analysis is to divide a structure into a finite number of nodes and elements and apply these physical principles to each and analyzing the loads of a node based on the loads on its adjacent nodes. The way this is done is via the following simplification of the basic equations mentioned previously.

$$F = \left(\frac{AE}{l_o}\right)\Delta l \tag{2}$$

where F is the force on an element, AE/l_o represents the stiffness of the element between two nodes and Δl is the displacement of a given node. The only unknown in the Equation 2 is the displacement which is what is determined by using ANSYS. Then from the displacements, the stress on each element can be solved for through Hooke's Law. Information on the equations used for stress and deformation calculation can be found in reference [11].

B. *Theory of Finite Element Heat Transfer Analysis*

The main relationship behind heat transfer analysis, whether it is one-dimensional, two-dimensional or three-dimensional, is:

$$\{H\} = [C]\{T\} \tag{3}$$

where {H} corresponds to the heat flux at each node, [C] represents the conductance matrix of all elements, and {T} represents the temperature of each node in a structure [10]. In many situations, the temperatures at either extreme of a structure would be known, or at least the desired temperatures would be known. Shape functions are computed to represent the temperature distribution in the tetrahedron. Equation 4 below represents a general equation for the temperature distribution for an element.

$$T = S_1T_1 + S_2T_2 + S_3T_3 + S_4T_4 \tag{4}$$

where S_1,S_2, S_3, and S_4 represent the shape functions of the element. The shape functions can be derived from conduction and convection equations which can be found in reference [12]. From Equation 4, the temperature at any point on the beam can be computed. Once all of the temperatures are known, and by knowing the thermal properties of the materials used, the heat flux on the structure can be computed from Equation 3.

C. *Fatigue Life*

Fatigue life is a concern in any device that undergoes cyclic loading. Industry commonly uses Stress versus number of cycles (SN) curves when determining fatigue life. SN curves are determined experimentally and already exist for most materials [13].

D. *Theory of Finite Element Modal Analysis*

Continuous systems theoretically possess infinite numbers of degrees of freedom and natural frequencies. However, for most practical applications, only the first few natural frequencies are important. Modal Analysis of a structure involves Newton's Second Law which simplifies to Equation 5 below:

$$-k\delta_{static} - ky + W = m\ddot{y} \tag{5}$$

Since $k\delta_{static}$ = W for the considered system, Equation 7 simplifies to the following:

$$m\ddot{y} + ky = 0 \tag{6}$$

By rearranging the variables the following equation can be obtained.

$$\ddot{y} + \omega_n^2 y = 0 \tag{7}$$

where $\omega_n^2 = k/m$, and is equal to the natural frequency squared [14]. As can be seen from the Equation 7, the natural frequency of a structure depends on the mass and the stiffness of the structure and is therefore unique to every structure.

In general, the governing equations of motions of continuous systems are partial differential equations whose exact solutions require both boundary and initial conditions. The solutions are often complex and difficult to find. For numerical approximation of discrete models, the matrix form of equations of motion for multiple degrees of freedom subjected to forces is Equation 8 from reference [10]:

$$[\boldsymbol{M}]\{\ddot{\boldsymbol{x}}\} + [\boldsymbol{C}]\{\dot{\boldsymbol{x}}\} + [\boldsymbol{K}]\{\boldsymbol{x}\} = \{\boldsymbol{F}\} \tag{8}$$

where $[\boldsymbol{M}]$is the mass matrix element, $[\boldsymbol{K}]$ is the stiffness matrix, and $[\boldsymbol{C}]$ is the damping matrix which is typically computed from a linear combination of the stiffness and mass matrices.

SIMULATION RESULTS

There were three different types of analyses modeled through ANSYS: Static, Modal, and Thermal. The Static Analysis involved applying the maximum allowable loading conditions on the device to assure that the stress levels and deflections were adequate for the application. The second type of analysis, the modal analysis, was performed to determine the natural frequency of the assembly and to ensure that there were no forcing functions, under normal operation, that would have frequencies near the structures natural frequency. Finally, the last analysis type used was thermal analysis. This type of analysis focused on the heat generation of the electric motor on the device and how the heat was dissipated throughout the structure. Since the device will be in contact with a person's skin, it is very important to understand the temperature distribution on the device and ensure that the temperature does not reach unsafe levels on any surface exposed to skin contact. The following sections describe in detail the results obtained from each analysis.

Static Load Results

From Figure 6 below, it can be seen that the stress levels of the knee loading device when applying 6.90 KPa of pressure on a person's knee are well within acceptable limits.

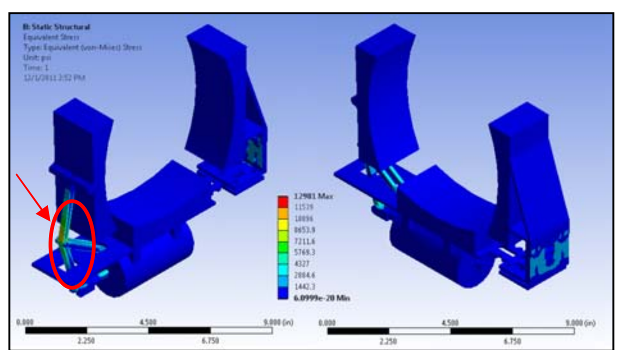

Fig 6.von-Mises stress contour plot.

As expected the stress is concentrated on the arms of the device. The maximum stress calculated along the lever arm is 89.5 MPa. From a structural strength point of view this equates to a 2.8 factor of safety, since the yield strength of steel used in the model for the device has yield strength of approximately 249.9 MPa. The next characteristic of the static analysis to consider is the total deflection of the arms as they apply the load to the patient's knee. As can be seen from Figure 7 below, the maximum deflection is 0.609 mm. However, this deflection takes place in the upper section of the application arm, and it is important to consider that the majority of the support to the knee will take place in the middle portion of the application arm. This section only deforms 0.279 mm, which corresponds to 4.4% of the axial motion which is an acceptable amount for this application.

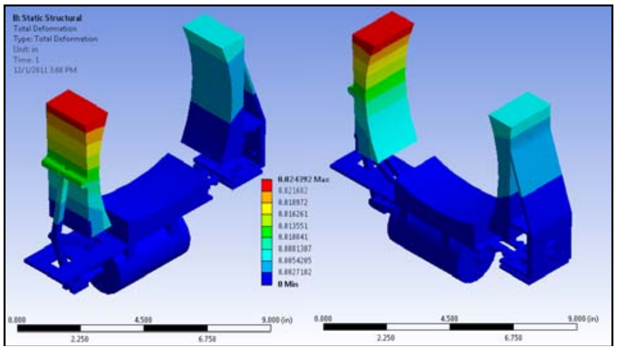

Fig.7. Total deformation of device.

The final factor to consider in the structural analysis was the fatigue life, by taking into consideration the duty cycle loading of the device. Figure 8 below shows the results for the fatigue life of the device.

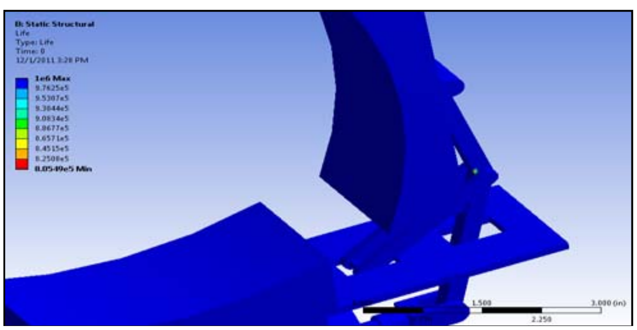

Fig.8. Cycle fatigue life.

The results from the fatigue analysis revealed that the device will last approximately 805,000 cycles. Given the duty cycle of 5 minutes of daily operation per patient with a 1 Hz frequency loading function during operation; or 300 cycles per operation, this device is designed to last for 2683 uses. The choice of 1 Hz as one example was linked to daily human physical activities such as walking. The device is able to induce loads up to 20 Hz, and it is a future task to evaluate appropriate loading frequencies.

Thermal Analysis Results

A major concern with any project involving motors is heat generation. For this project, this too was important to analyze. Since this device will be in direct contact with theuser, it is crucial that the device will operate at a comfortable temperature on all the accessible surfaces. Considering these factors, the internal heat generation of the motor was assessed. To accompany this data, the convection coefficient for the metal surfaces needed to be derived, since they are in contact with the surrounding air. The values for the heat generation and convection coefficient were 0.9688 W/cm^2 and 0.05258 W/(m^2*K) respectively. Shown below in Figure 9 is the first transient thermal analysis that was performed.

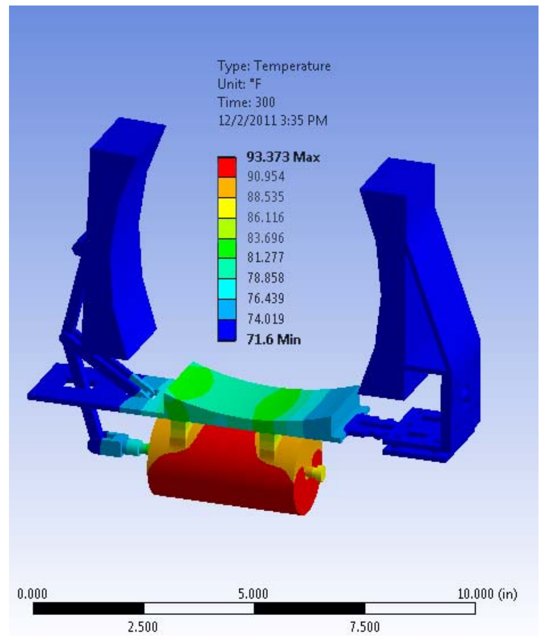

Fig.9. Initial design.

Although the temperature of the motor is calculated to be approximately 34.4°C, this is not detrimental because this will be enclosed and not accessible to the user of the device. However, there is some concern with the temperature of the bottom polyethylene pad. The temperature of this pad increases to approximately 28.9°C after 300 seconds of operating time. While this is not a hot temperature, it is undesirable because the user of this device may feel this temperature change. In order to resolve this issue, two different viable options were proposed; to change the material of the pad, or to increase the thickness of the pad. The design improvements for this device will be discussed subsequently.

Design Improvements

The design process involves multiple iterations. Thanks to modern technology this process, as well as the product development process, can be streamlined in a quick inexpensive manner using CAD/CAM/CAE systems. The most common method involves creating the geometry in a CAD system, analyzing the model for potential failures, redesigning the model to compensate flaws, and re-analyzing the new model. This process can be followed many times until a satisfactory design is found. Then the part can be prototyped and physically tested, thus saving much time and money. For this project, the process previously described was used to find the optimal design solution. The device needed to be redesigned for increased strength and lifetime as well as decreased pad temperature.

After conducting the static structural analysis the maximum von-Mises stress was approximately 89.5 MPa. With the yield strength of steel being 249.9 MPa, our design has a factor of safety of 2.79. A factor of safety of approximately 3 can be considered adequate in most cases; however, because this device cycles every second, a larger factor of safety can help improve fatigue life. Common industrial practice states $1x10^6$ cycles as infinite life. The original design had a lifetime of $8.05x10^5$cycles. This equates to 2682 usages. With a feasible clinical usage of multiple usesper day, the device could potentially fail in two to three years. Thus, a redesign was required on the lever arm, because this is where the highest stresses occurred and was where ANSYS determined the device would fail first. The bend in the lever arm was the weakest point so a chamfer was added to strengthen the arm

After completing the new geometry the analysis was again performed to verify that the stress was indeed lowered and the life was increased. Figure 10 below shows the maximum stress of the device to be 47.4 MPa.

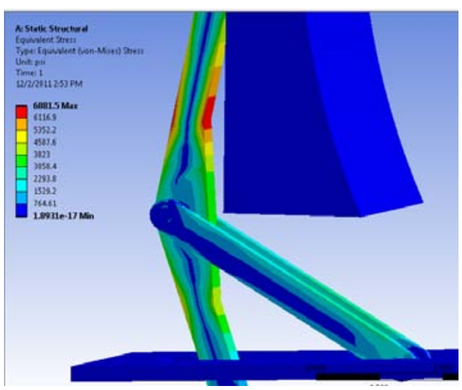

Fig.10. Von-Mises stress in redesigned device.

With the new lower operating stress the factor of safety becomes closer to 5.25. The lower stress and increased factor of safety significantly contributes to lifetime longevity. The entire device including the lever arm is now rated at an infinite lifetime.

Another critical aspect of this design was the heat generated by the electric motor. Normal operation involves the motor continuously powered for five minutes. With no forced convection the heat can built up rather quickly reaching a temperature of 33.9°C on the motor. Since the motor itself will be protected, the surface that makes contact with the leg is the only critical surface to monitor. The maximum temperature reached on this surface was found to be approximately 28.9°C. Although this temperature is a safe-to-touch temperature, there is a potential for discomfort, which is unacceptable for this design.

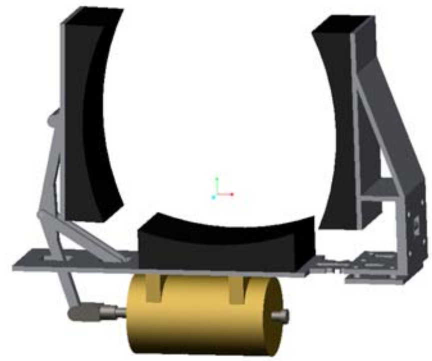

Fig. 11.Final design with thicker pad.

Thus, there were two available options; either alter the materials used for the pad, or increase the thickness of the pad. The pad is currently made from polyethylene, which already has a rather low thermal conductivity, so it was decided to increase the thickness of the pad. Figure 11 above shows the new design with a thicker pad.

The pad's thickness was increased from 12.7 mm to 25.4 mm. This will not only decrease the heat transfer, but it will also increase the comfort of the user. In the figure below, the temperature on the pad's surface is now a more comfortable 23.9°C.

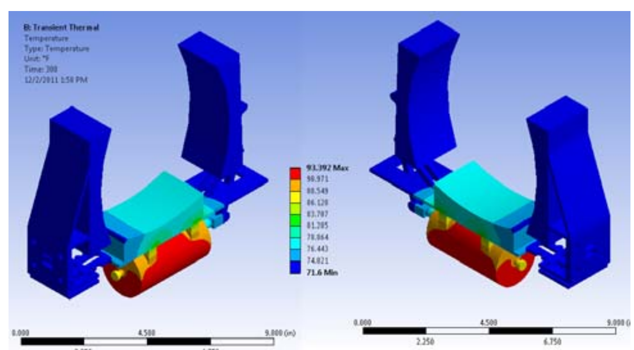

Fig. 12.Final design.

One final issue that was addressed was what the temperature would reach if the device were operated beyond a 300 second operating cycle. While the device was designed to operate successfully for 300 seconds, it is a possibility that the device could be used or left powered-on longer than the designed time. For this issue, a time versus temperature plot is useful (Figure 13).

After analyzing the generated results from ANSYS, it became apparent that at approximately 300 seconds, the device achieves steady-state temperature conditions. This result is positive because the device can safely be used, or left on for a long period of time, and would notfail or cause damage to the environment or the user.

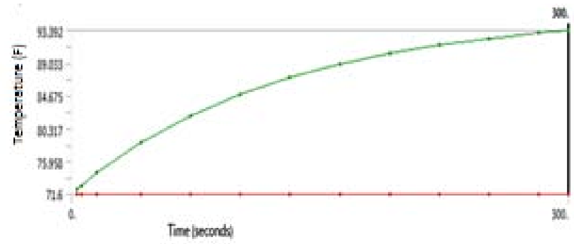

Fig.13. Temperature versus Time.

Modal Analysis Results

Primary design requirements for electromechanical devices are stability, strength, and the dynamic behavior in operation stage; which can be achieved by producing a structure with a high stiffness, by keeping dimensional tolerances small, and by minimizing the vibration loading in the operation stage. However, estimates alone are not sufficient to perform a reliable and precise evaluation of the stability of such a device. The dynamic forces exerted between the linkages, pins, and the electric motor may produce faults and/or dynamic behavior resulting in more deformation, rubbing, and possibly interferences. Modal testing, as shown in Figure 14, can provide valuable information about the dynamic response characteristics. This information typically includes the natural frequencies and mode shapes of the simple and outer geometry shapes at room temperature which can serve as a valuable quality control test.

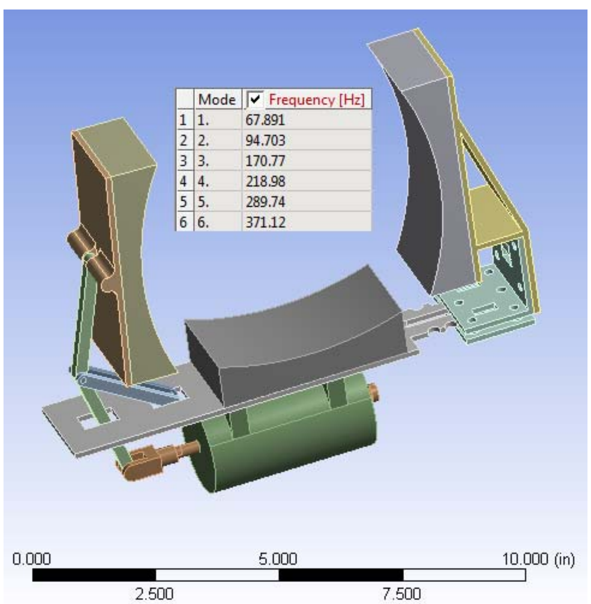

Fig. 14.Device with analysis of 6 modes and corresponding frequencies.

The basic criteria which determine the dynamic response levels are: proximity to resonance, excitability of the mode, and force and damping levels. Dynamic response characteristics during operation are likely to differ significantly from those measured under the modal test condition.

CONCLUSIONS

The deformed geometry, stress distribution, modal frequencies, temperature distribution, and life expectancy of a knee loading device were analyzed with ANSYS workbench. The device was modeled using steel parts. Although the stress levels were considerably low, the fatigue life was examined (since the duty cycle of the device is deals with cyclic loading conditions). Analysis demonstrated that added rigidity to the application arm significantly increases the life of the device. When examining the thermal distribution, the pad on which the user's leg rests was determined to become hotter than desired. In order to lower the temperature of the pad surface the pad was thickened. This was found to have the desired effect of significantly decreasing the temperature on the pad's face. The modal frequencies were found to be much higher than the operating range of the device, verifying that they will not be a factor in its operation.

The durability concerns that were found were successfully addressed, with the thickening of the base pad to insulate against the motor's heat generation and the increased life expectancy of the device from the added stiffness to the application arm, so that the final device will operate as required.

ACKNOWLEDGMENT

This work was made possible through the past and current research of the laboratories of Mechanical and Biomedical Engineering Departments, IUPUI.

REFERENCES

[1] Zhang, Ping, Kazunori Hamamura, Hiroki Yokota, and George Malacinski (2009). "Potential Applications of Pulsating Joint Loading in Sports Medicine." *Exercise and Sport Sciences Reviews* 37:52-56.

[2] Smith & Nephew Corporate. (2011). *ultrasound bone healing system*. Retrieved from http://global.smith-nephew.com/master

[3] *Physio-stimlite*. (2011). Retrieved from http://www.orthofix.com/ intl/contact/distrib/physiostimupper_frame.htm

[4] Anthem Inc., "Electrical Bone Growth Stimulation." DME.00004. 10/12/2011. *Anthem Medical Policy*.Web. 21 Jun 2012. <http://www.anthem.com/medicalpolicies/policies/mp_pw_a050280.htm>.

[5] Anthem Inc., " Ultrasound Bone Growth Stimulation." DME.00004. 10/12/2011. *Anthem Medical Policy*.Web. 21 Jun 2012. <http://www.anthem.com/medicalpolicies/ policies/mp_pw_a050287.htm>.

[6] Zhang, P., Turner, C.H., Yokota, H. (2009). *Joint loading-driven bone formation and signaling pathways predicted from genome-wide expression profiles*. Bone 44:989-998.

[7] Zhang, P., Su, M., Liu, Y., Hsu, A., and Yokota, H. (2007). *Knee loading dynamically alters intramedullary pressure in mouse femora*. Bone 40:538-543.

[8] Kwon, R.Y., Meays, D.R., Tang, W.J., Frangos, J.A. (2010). *Microfluidic enhancement of intramedullary pressure increases interstitial fluid flow and inhibits bone loss in hindlimb suspended mice*. J. Bone Miner. Res. 25:1798-1807.

[9] ANSYS Inc. (April 2009). Engineering data.*Release 12.0*, Retrieved from http://www1.ansys.com/customer/content/documentation/120/wb_eda.pdf

[10] Moaveni, Saeed. *Finite Element Analysis: Theory and Application with ANSYS*. 3rd ed. Upper Saddle River, NJ: Pearson Prentice Hall, 2008.

[11] Beer, Ferdinand P., Johnston, E. R. *Mechanics of Materials*. 5th ed. Avenue of the Americas, New York, NY: McGraw-Hill, 2009.

[12] Incropera, Frank P., DeWitt, David P. *Fundamentals of Heat and Mass Transfer*. 6rd ed. River Street, Hoboken, NJ: John Wiley and Sons, 2007.

[13] Smith, William F., Hashemi, Javad. *Foundations of Materials Science and Engineering*. 4th ed. Avenue of the Americas, New York, NY: McGraw-Hill, 2006.

[14] Ogata, Katsuhiko. *System Dynamics*. 4rd ed. Upper Saddle River, NJ: Pearson Prentice Hall, 2004.

[15] Pelmear PL & Wasserman DE. *Hand-Arm Vibration: A Comprehensive Guide for Occupational Health Professionals.* 2nd ed. Boston: OEM Health Information Inc., 1998.