A FINITE ELEMENT APPROACH TO ANALYZE HUMAN KNEE JOINT REPLACEMENT PROSTHESIS

Conference Paper · July 2020

Pathalamuthu Pitchaimuthu
Anand Institute of Higher Technology

A FINITE ELEMENT APPROACH TO ANALYZE HUMAN KNEE JOINT REPLACEMENT PROSTHESIS

P.Pathalamuthu[1] Dr.J.Selwin Rajadurai[2]

1. P G Scholar, Engineering Design,
   Government college of Engineering, Tirunelveli.
   e-mail: muthup2001in@yahoo.co.in
2. Senior Lecturer, Mechanical Engineering Department,
   Government college of Engineering, Tirunelveli.

ABSTRACT:

In the past decades, the life expectancy of the population has increased substantially, a fact that has led to an increase in the prevalence of degenerative joint diseases. The knee joint is the largest joint in the human body. Although commonly called a 'hinge joint', it is for more complex than that. Bending of the knee takes place through a combination of rolling, gliding and rotation of the bones that makes up the joint. The common causes of knee pain and loss of knee function in clinic are osteoarthritis, rheumatoid arthritis and post traumatic arthritis. When the knee joint is damaged due to above diseases, it must be replaced for prosthesis. The finite element method (FEM) one of the most advanced simulation technique in solid mechanics is used for orthopedic biomechanics. It is used as a tool for design and analysis of total joint replacement. The purpose of the study is to investigate the behavior of newly designed implants under body weight loads during stumbling and impact loading by parametric models (Solid Edge) and analysis (ANSYS 11). Engineers and Orthopedics combine to make a person’s life a normal painless life.

Keywords: Finite Element Method, Human Knee joint, Prosthesis, Impact loading and Fatigue Analysis

1. INTRODUCTION:

The knee is the largest and most complex joint within the human body, consisting of both the femoropatellar joint, between the patella and the femur, and the tibiofemoral joint, between the femur and tibia as shown in figure 1. Proper motion of the joint relies significantly on the function of soft tissue constituents including the four ligaments of the tibiofemoral joint: anterior and posterior cruciate ligaments (ACL and PCL respectively) and medial and lateral collateral ligaments (MCL and LCL respectively).These ligaments allow primarily flexion/extension and rotation of the joint by enabling the bony constituents (femur and tibia) to translate and rotate relative to each other. In addition to ligaments, soft cartilage in the joint space permits nearly frictionless contact between the bones. Orthopedic implants are intended to support forces and must thereby be firmly attached to the rest of the skeleton cemented implants is still used in the majority of operations.

Figure1- Human Knee joint

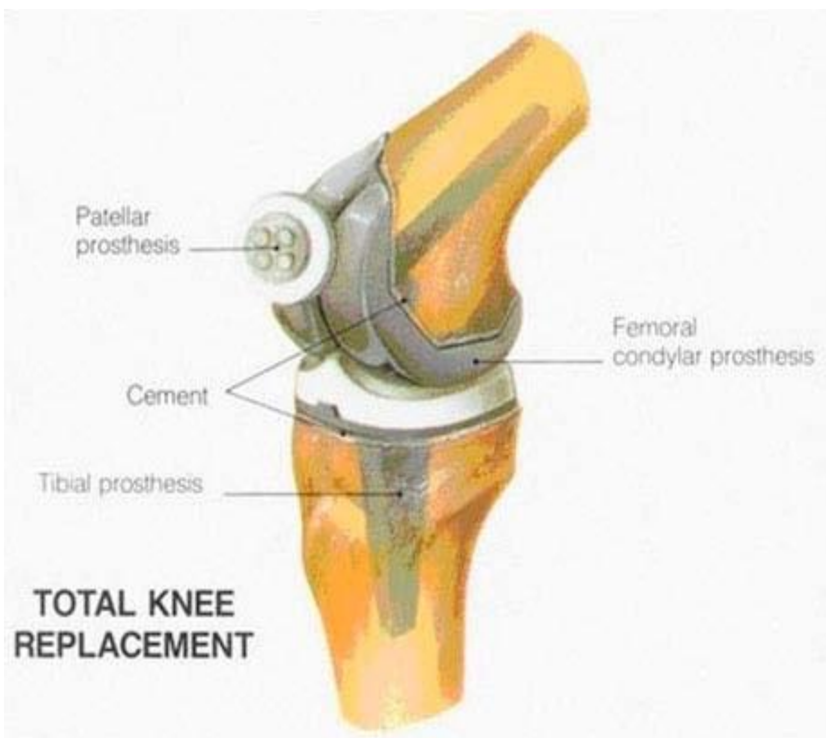

Figure 2-Human Knee implant

Conventional design and analysis of bone implant knee prosthesis rely on expert's knowledge, experience and ability to avoid any irreversible damage on the bones of patients. Because of the difficulty of performing implant tests in vivo, mathematical models have been developed to carry out the structural analysis of implants before application on a patient. Accordingly bone implant knee prosthesis could be designed and studied with computer simulation. The Finite Element Method (FEM) is an advanced simulation technique that has been used in orthopedic biomechanics. It is an important tool used in the design and analysis of total joint replacement and other orthopedic devices.FEM modeling and analysis present a nondestructive design approach for bone implant knee prosthesis. It allows many what if scenarios to be studied in computer environment before the prosthesis is actually inserted. This simulation streamlines the design and prevents any permanent damage caused by misimplementation.

2. COMPONENTS OF TOTAL KNEE PROSTHESIS:

There are three primary components in total knee implants. These include the femoral, tibial and patellar components as shown in figure 2.The femoral component is typically made of metal called cobalt chrome (co-cr). Co-Cr is a very hard and durable material allowing it to withstand the massive loads and cycles. The tibial component is typically made of Titanium alloy. The intermediate layer meniscus is made out of Polyethylene. The cement material used to join tibial tray and meniscus is made from

Polymethylmethacrylate (PMMA). The material properties of these components are shown in the following table.

Table 1: Material Properties

Material	Young's Modulus(MPa)	Poisson's Ratio	Yield Strength(MPa)	Tensile Stength(MPa)
Titanium Alloy	96,000	0.36	830	900
Polyethyle ne	32,000	0.2	27	47
PMMA	2,500	0.38	110	110

3. ANALYSIS:

3.1 Finite Element Method

The Finite Element Method (FEM) is a computer aided mathematical technique for obtaining approximate numeral solutions to the equations of calculus that predict the response of physical systems subjected to external influences. The general steps involved to solve a problem using FEM can be summarized as follows:

1. A problem domain is divided into regions called elements. The collection of elements is called a mesh.
2. The properties and the governing equations are assumed over than elements and expressed mathematically in terms of unknown values at specific points in the elements called nodes.

3. An assembly process is used to link the individual elements to the given system.
4. When the effects of loads and boundary conditions are imposed a set of algebraic equations is obtained.
5. Solutions of these equations give the approximate behavior of the continuum.
6. The continuum has an infinite number of degrees of freedom (DOF), while the discretized model has a finite number of DOF. This is the origin of the name finite element.
7. The approximation of the continuum domain with finite elements provides good precision. Simply increasing the number of elements can achieve increasing precision.ANSYS package is used for the current problem.

3.2 Modeling and Meshing:

The assembled knee prosthesis is converted as parasolid so as to have compatible with ANSYS workbench.

Figure 3- 3D model of Knee prosthesis

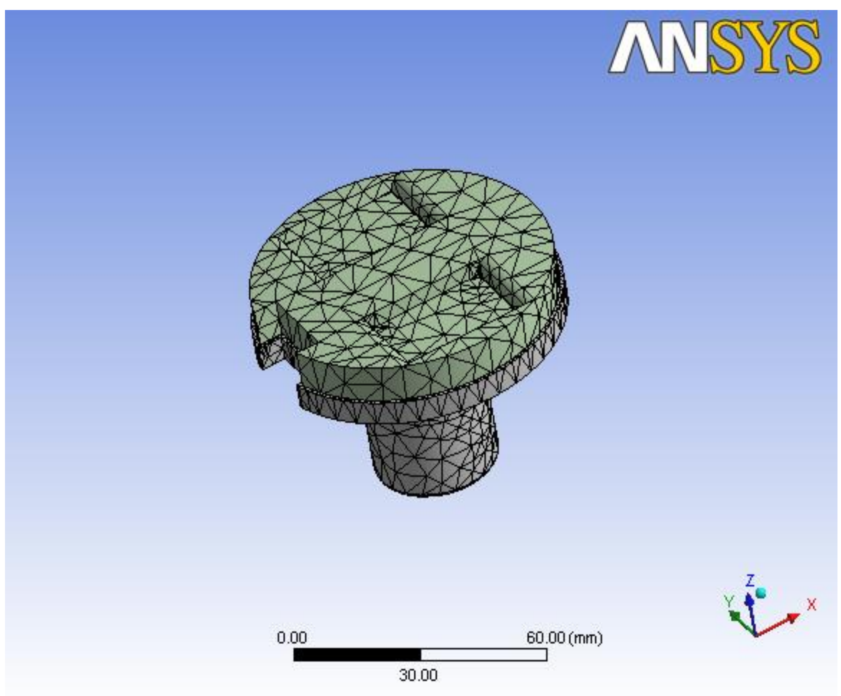

Figure 4-Meshed model of knee prosthesis

Thus formed 3D geometric model in ANSYS workbench is shown in figure 3. Geometric shapes have big influence on the performance of prosthesis. Those with smooth surfaces generally reduce stress concentrations and lead to high fatigue life of prosthesis. Those with sharp of non smooth surfaces provide good bonding capability at the interface and prevent possible sliding at the interface. The level of stress concentration and tendency for fatigue failure depend on the sharpness of the stem surfaces. In this study, a prosthesis geometry that will yield low stress values and high fatigue life is investigated. The generated model is meshed using solid185 element is shown in figure 4. Total number of elements generated in the mesh is 10,568 with 22,066 nodes. A typical meshed Knee prosthesis is shown in figure 4.

3.3 Results and Discussions:

The lower part (tibial stem) of the model is constrained in all directions, while the applied load was normally distributed on the femoral contact area of the polyethylene layer (Meniscus).A load of 500N was applied on each femoral contact area of the polyethylene layer.

Deformation:

The maximum deformation occur at the outer end of the meniscus shown in figure5.These deformation within the safe limit so,our design is safe.

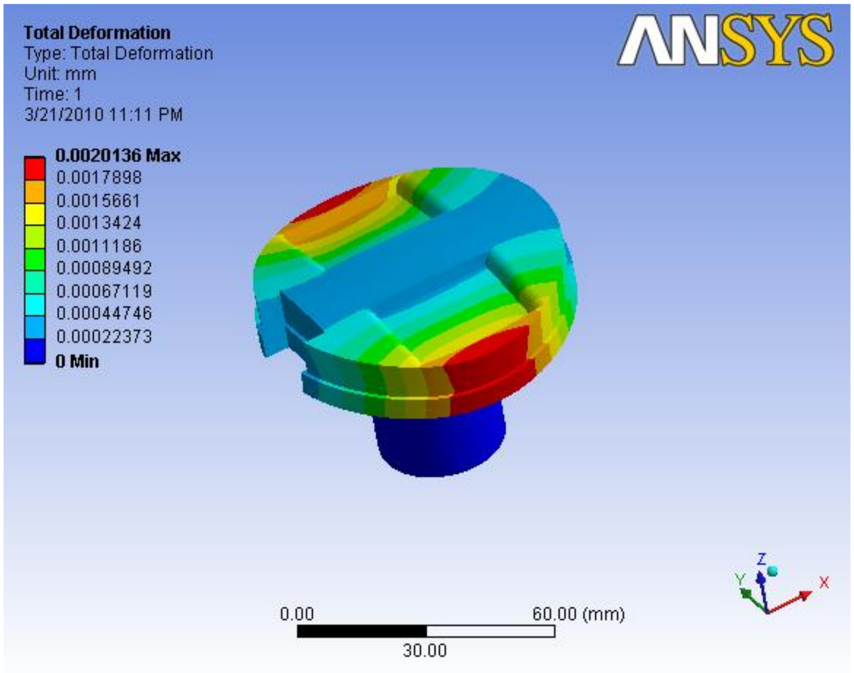

Figure 5- Deformation

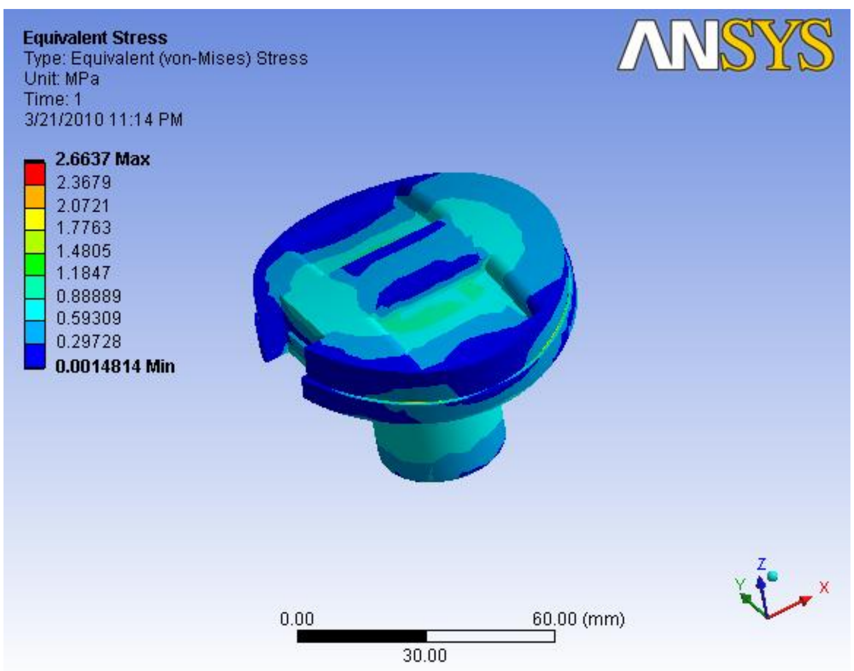

Figure 6-Von Mises stress

Von Mises stress:

The Von mises criterion states that failure occurs when the energy of distortion reaches the same energy for yield/failure in uniaxial tension. Mathematically, this is expressed as

$$\sigma_v = \sqrt{\frac{1}{2}}\left[\sigma_x - \sigma_y\right]^2 + \left[\sigma_y - \sigma_z\right]^2 + 3\left[\tau_{xy}{}^2 + \tau_{yz}{}^2 + \tau_{zx}{}^2\right]$$

4. CONCLUSION:

This study is used to analyze the newly designed prosthesis (stress analysis, deformation analysis) to avoid breakage, wear, etc.It shows the static FE analysis of TKR was conducted using ANSYS.

A general conclusion is that implant knee prosthesis can be designed and studied with computer models before implementation on the patient. This procedure reduces design time while helping to prevent permanent damage caused by miss implementation.

5. WORK COTINUATION FIELDS:

Fatigue Analysis, Impact Analysis

REFERENCES:

1. **Oguz Kayabasi, Bulent Ekici** (August 2006) - The effects of static, dynamic and fatigue behavior on three dimensional shape optimization of hip prosthesis by finite element method.-.p1-9

2. **A.Al-samhan, S.M.Darwish, H.Al-Khawashgy** (Sep 2006)-Optimization of polyethylene layer thickness of artificial knee

3. **Jessica N. Albino Soto, Nitza Correa Cora and Jessenia Feliberty Irizarry**-Mechanics of Biomaterials: Orthopedics.p8, 9

4. **Dr.Thomas, Dr.B.L.Prakash-** HOSMAT Joint Replacement center, HOSMAT Hospital-Bangalore

5. **Junifen shi** (June 2007) - Finite Element Analysis of Total Knee Replacement considering Gait cycle load and malalignment.

6. **Sara Checa PhD**-Finite Element Modeling in Biomechanics.