American Journal of Psychiatric Rehabilitation

Expert Opinion Article

Impact Analysis Of Femoral Implant Of Injured Human Knee Using Finite Element Analysis

Dr. Raj Kumar.E[1*], Dr.Bindu.SS[2], Anju Vikraman VJ[3], Manoj.A[4], Niju V S[5], Krishnakumar.K[6], Sree Raj.M.P[7], Joe Jeba Rajan.K[8]

[1]Professor, Department of Mechanical Engineering, NET Engineering and Technology, Rajkumarpillai2020@gmail.com
[2]Associate Professor, Department of Mechanical Engineering, Rajadhani Institute of Engineering and Technology,
[3]Assistant professor, Vidya college of Engineering
[4,5,6,7,8]Assistant Professor, Department of Mechanical Engineering, Rajadhani Institute of Engineering and Technology,

***Corresponding Author:** Dr. Raj Kumar.E
*Email: Rajkumarpillai2020@gmail.com

Abstract:
The impact analysis of the artificial femoral component and optimization is a crucial aspect in the design and development of hip replacement surgeries. Analysis involves studying the forces and stresses that are applied on the artificial femoral component during different activities, such as walking, running, and jumping. By understanding these forces and stresses, we can optimize the design of the artificial femoral component to ensure its durability and longevity. This is especially important as hip replacement surgeries have become more common among aging populations. A replacement surgery is used to enhance the quality of life for patients. Additionally, it can also reduce the risk of complications and potential revisions, ultimately leading to cost savings in the long term. This paper mainly concentrates on impact analysis of the artificial femoral component and optimization.

Key words: Knee implant, 3D model, Impact analysis, Biomechanics.

1. Introduction:

The human knee is working under variable fatigue stresses in our today's normal life. This is a most stressed part of a human anatomy a very complicated evolution in human body. The knee of a human being is having an essential role in the transport of people as shown in figure.1. The impact of femoral implants on injured human knees using advanced modeling and simulation techniques has greatly improved the outcomes of knee injuries.

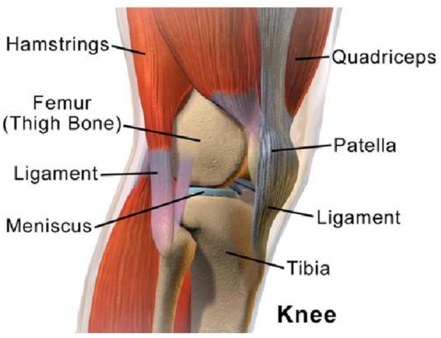

Figure1. The anatomy of human knee

The human knee in recent days suffer a lot of problems including tearing of the meniscus, breaking or injuries of knee bones including patella, femur and tibia as shown in figure.2. The external aids for the knee injuries always found to be unfair solution of the normal life of the patients.

Figure 2. Knee anthroplasty

The first knee implant is performed in 1968. Over the years, the process of knee anthroplasty is evolved a lot in the last two decades. The application of direct printing of components through 3 d printing has attained its place in the medical field. It is used in medical field due to the faster operation and flexibility in each component design. Still the solution has not reached a comfort and affordable level for the patients. There are many problems in the process.

The major problem persisting are

- The human knee design is totally different from one to another human in microscopic and nanoscopic level.
- The strength gap between the human bone and the implant.
- The failure of implants inside the patient body complicates the process of the operating the patient.
- Losing of normal life of the patient by torturing implant inside body.

2. Literature review

The success rate of the implant is not getting over 5 years for 1 of 10 patients got the implants and not more than 10 years for 2 out of 10 patients. Most of the treated patients have a pain during rising up from the sitting position and during climbing stairs. The finite element method of analysis is a useful tool to calculate stress and strain in complex systems. So the analysis is carried out on this process by number of researchers for the femoral component in past two decades. The failure of polyethylene implants inside the patient body is a major failure in the knee anthroplasty. The removal of the broken components from the patient's body becomes complicated than the implant surgery. The broken tissues become more complicated because of infections. The major issues in the earlier researches are the model created in the CAD software by getting the dimensions of the knee and adjusting for the component for that size. The actual knee model is not as an accurate in the model created by the engineers by the dimensions of outer shape of the patient. So the advantages of the reverse engineering of this decade are applied to create the model of the human knee. The scanned model of the human knee is used for the creation of 3D model of the same. The factors affecting the implant life are BMI, age, BMD, occupation and lifestyle. These factors making difference in the impact of dissimilar knees and implant assembly. The impact analysis is becomes necessary in this situation for each patient.

3. Methodology

The optimization process involves using computer simulations and prototypes to test different materials, shapes, and sizes of the artificial femoral component to find the most suitable design for maximum strength and stability. MRI scan of the human knee is capture to create the 3D image. Reverse engineering technique is applied for convert the 3D image into 3D model. The femoral component of knee is designed using the 3D model in Solid works. The model is then analyzed for impact with the help of Ansys workbench and is optimized. The research methodology adopted in this paper is shown in figure.3.

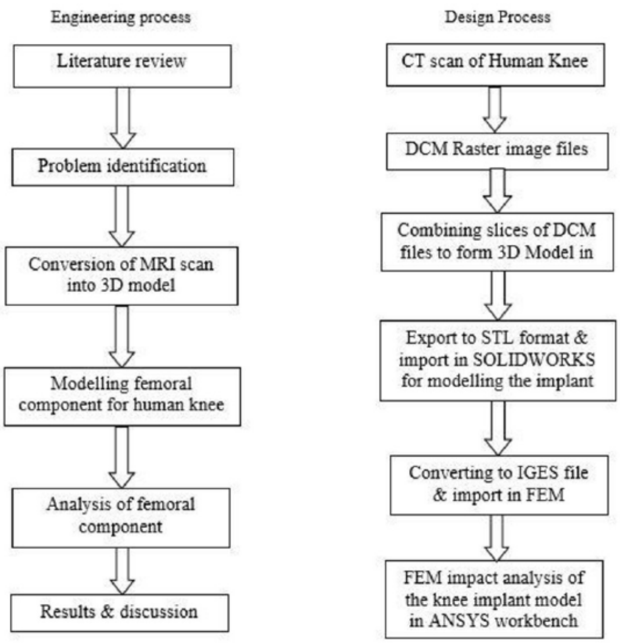

Figure 3. Methodology

4. CAD Modeling

The femoral component of the knee is a crucial part of the overall design of a knee replacement. In order to ensure its effectiveness and longevity, it is important to create a precise and accurate design. This is where the use of 3D modeling in Solid Works comes into play. By using this advanced software, designers are able to create a detailed and realistic model of the femoral component, taking into account every intricate detail and dimension. Once the 3D model is complete, it is then analyzed using Ansys workbench to simulate different impact scenarios. This allows for a thorough assessment of the design's strength and durability. Any potential weaknesses or areas for improvement can be identified and addressed before the actual production of the femoral component.

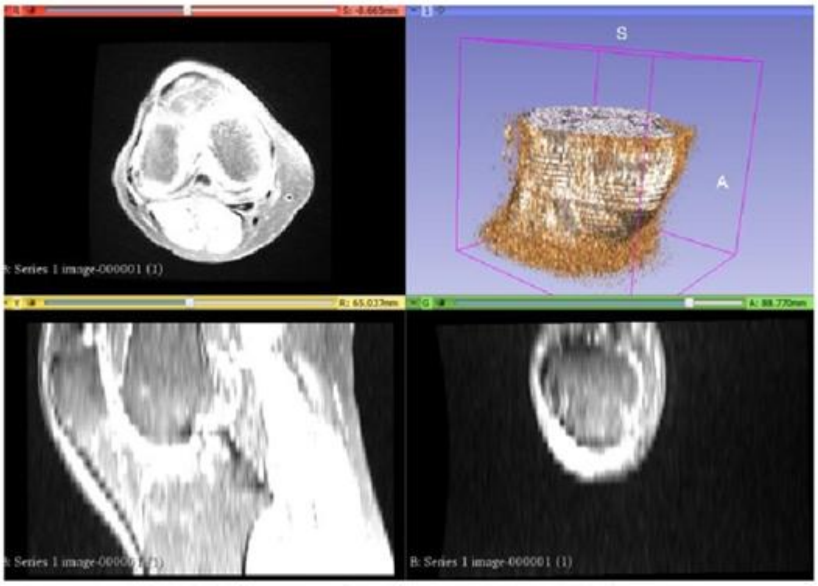

Figure 4. Conversion knee model into .stl format

Conversion of a knee model into .stl format is a process that involves transforming a 3D model of a knee joint into a file format that is compatible with 3D printing as shown in figure 4. This conversion is necessary for creating physical replicas of the knee model for medical or educational purposes. The process begins with obtaining a high-quality 3D model of the knee from medical imaging techniques such as MRI or CT scans. The model is optimized using specialized software to remove any imperfections and ensure smooth surfaces and accurate dimensions. Next, the model is exported into .stl format, which is widely used in 3D printing due to its ability to represent complex geometry. The final .stl file can be used to print a physical model of the knee using a variety of 3D printing techniques, making it an essential step in bringing virtual models to life for practical use in the medical field. Overall, the conversion of a knee model into .stl format plays a crucial role in bridging the gap between virtual and physical models, allowing for more practical applications in the medical industry.

5. Analysis

The IGES (Initial Graphics Exchange Specification) model is a standardized file format used for transferring 3D models between different CAD (Computer-Aided Design) systems. Once a model is created and saved in IGES format, it can be easily imported into other software programs for further analysis and manipulation as shown in figure.5. This capability makes IGES an essential tool for collaboration and data exchange in the design and engineering industries. In particular, the IGES model is commonly imported into the ANSYS workbench, which is a powerful simulation and analysis software used for various engineering applications. By importing the IGES model into ANSYS workbench, engineers and designers can utilize its advanced tools and features to perform detailed finite element analysis, structural simulations, and other types of engineering analyses. This seamless integration between IGES and ANSYS workbench makes it easier for professionals to accurately assess the performance of their designs and make informed decisions to improve their product or design. Overall, the import of IGES models into ANSYS workbench plays a crucial role in enhancing the efficiency and accuracy of the engineering design process.

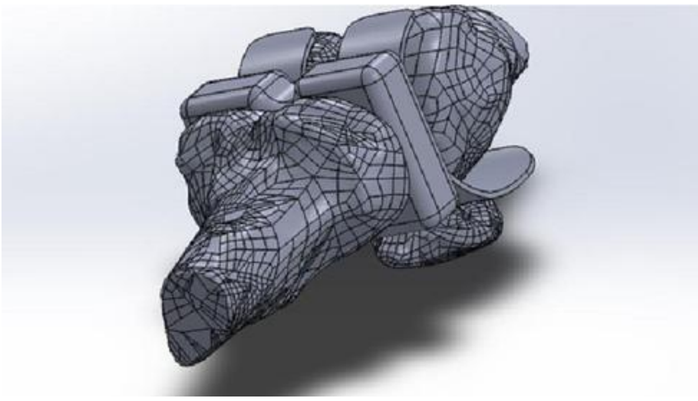

Figure 5. Knee model with femoral component

The ability to import patient-specific geometries from medical imaging (CT, MRI) allows for highly personalized designs, which can lead to better fit, alignment, and long-term outcomes. Ansys Workbench provides tools for parametric analysis and optimization. Engineers can easily vary design parameters (e.g., peg geometry, material thickness, shape) and quickly analyze their impact on performance metrics (stress levels, deformation, fatigue life). This allows for iterative design improvements and the identification of optimal designs.

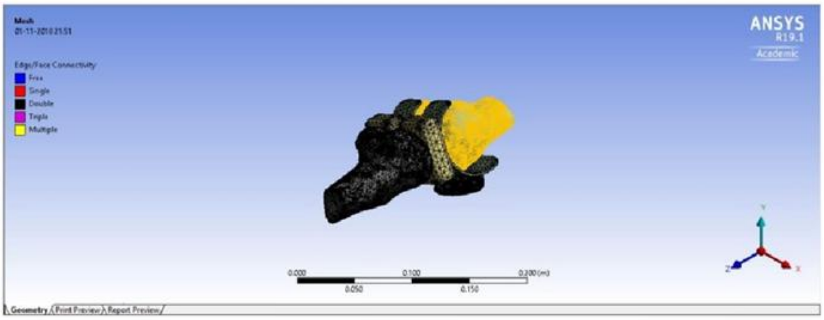

Figure 6. Knee model with femoral component

Simulating the intricate contact mechanics between the femoral component and the tibial insert, including friction, wear, and pressure distribution. This helps in designing components that minimize wear and improve longevity.

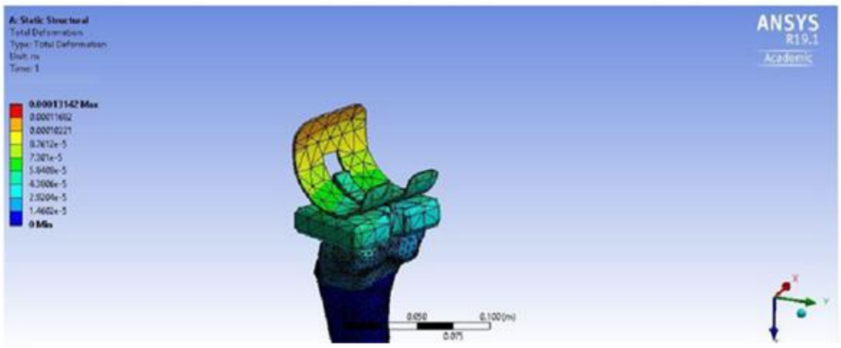

Figure 6. Knee model with Titanium component

Incorporating realistic material properties for both the implant (e.g., Co-Cr alloys, titanium alloys, ceramics) and the bone (which can be heterogeneous and anisotropic) as shown in figure. 6. Ansys can analyze both linear and non-linear material behaviours.

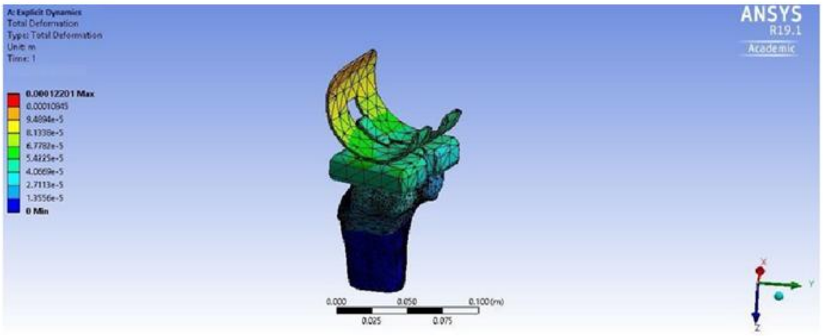

Figure 6. Knee model with Stainless steel

Accurately predicting stress distribution within the implant and the surrounding bone under various loading conditions (walking, running, bending, etc.). This is crucial for identifying potential failure points, optimizing material usage, and preventing issues like stress shielding (where the implant carries too much load, leading to bone degradation).

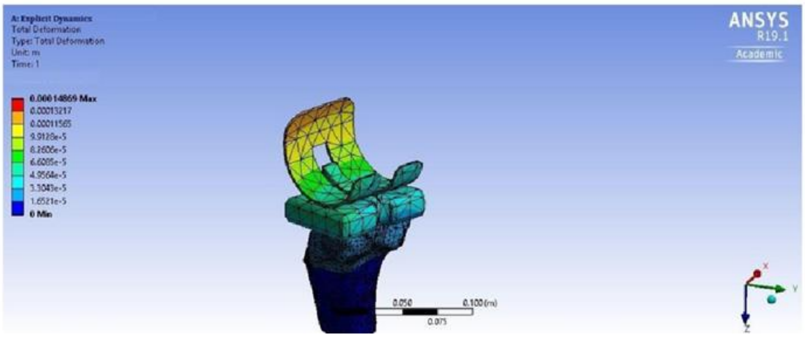

Figure 6. Knee model with Co-Cr-Mo alloy

Table.1 Total deformation of femoral component.
Through this process, the design can be optimized to ensure maximum functionality and safety for patients who will undergo knee replacement surgery using this component. The combination of 3D modeling and simulation with Ansys workbench results in a highly accurate and efficient approach to designing the femoral component of the knee.

Total deformation of femoral component					
Stainless steel		Titanium Alloy		Co-Cr-Mo alloy	
Minimum	0. m	Minimum	0. m	Minimum	0. m
Maximum	1.22e-003 m	Maximum	1.3141e-003 m	Maximum	1.4868e-003 m
Average	4.5594e-004 m	Average	4.7969e-004 m	Average	5.179e-004 m

Conclusion

By incorporating advanced modeling and simulation techniques, it is possible to create more accurate and personalized femoral implants, tailored to the individual patient's specific injury. This not only improves the fit of the implant but also allows for a more natural range of motion in the knee joint. Additionally, these techniques help to reduce the risk of complications and improve the longevity of the implant. With the use of computer-aided design and 3D printing, the production of femoral implants has become more efficient and precise, leading to better outcomes for patients. Furthermore, these advanced techniques also allow for virtual testing and adjustments before the actual surgery, minimizing the risk of errors and improving overall success rates. Analysis and optimization play a significant role in improving the success rate of hip Therefore, continuous research and development in this area are crucial for further advancements in hip replacement surgeries. The injured or teared human leg knee is operated by implantation of the artificial femoral component to aid walking for affected people. Overall, the impact of using advanced modeling and simulation techniques in femoral implants has revolutionized knee injury treatment, providing better outcomes for patients and improving their quality of life.

References:

1. Abdel-Rahman, E. M. , and Hefzy, M. S. , 1998, “Three-Dimensional Dynamic Behaviour of the Human Knee Joint Under Impact Loading,” Med. Eng. Phys., 20(4), pp. 276–290.
2. Anderson, F. C. , and Pandy, M. G. , 2001, “Dynamic Optimization of Human Walking,” ASME Journal of Biomechanical., 123(5), pp. 381–390.
3. Adouni, M. , Shirazi-Adl, A. , and Shirazi, R. , 2012, “Computational Biodynamics of Human Knee Joint in Gait: From Muscle Forces to Cartilage Stresses,” Journal of Biomechanical., 45(12), pp. 2149–2156.
4. Beillas, P. , Papaioannou, G. , Tashman, S. , and Yang, K. H. , 2004, “A New Method to Investigate in Vivo Knee Behavior Using a Finite Element Model of the Lower Limb,” Journal of Biomechanical, 37(7), pp. 1019–1030.
5. Baldwin, M. A. , Clary, C. W. , Fitzpatrick, C. K. , Deacy, J. S. , Maletsky, L. P. , and Rullkoetter, P. J. , 2012, “Dynamic Finite Element Knee Simulation for Evaluation of Knee Replacement Mechanics,” Journal of Biomechanical, 45(3), pp. 474–483.
6. Donahue, T. L. , Hull, M. L. , Rashid, M. M. , and Jacobs, C. R. , 2002, “A Finite Element Model of the Human Knee Joint for the Study of Tibio-Femoral Contact,” ASME Journal of Biomechanical. Eng., 124(3), pp. 273–280.
7. Li, G. , Gil, J. , Kanamori, A. , and Woo, S. L. , 1999, “A Validated Three-Dimensional Computational Model of a Human Knee Joint,” ASME Journal of Biomechanical. Eng., 121(6), pp. 657–662.
8. Limbert, G. , Taylor, M. , and Middleton, J. , 2004, “Three-Dimensional Finite Element Modelling of the Human ACL: Simulation of Passive Knee Flexion With a Stressed and Stress-Free ACL,” J. Biomechanical, 37(11), pp. 1723–1731.
9. Majewski, M. , Susanne, H. , and Klaus, S. , 2003, “Epidemiology of Athletic Knee Injuries: A 10-Year Study,” Knee, 13(3), pp. 184–188.
10. Penrose, J. M. , Holt, G. M. , Beaugonin, M. , and Hose, D. R. , 2002, “Development of an Accurate Three-Dimensional Finite Element Knee Model,” Computational Methods Biomechanical Biomedical. Eng., 5(4), pp. 291–300.
11. Pena, E. , Calvo, B. , Martinez, M. A. , and Doblare, M. , 2006, “A Three-Dimensional Finite Element Analysis of the Combined Behavior of Ligaments and Menisci in the Healthy Human Knee Joint,” Journal of Biomechanical, 39(9), pp. 1686–1701.
12. Wall, S. J. , Rose, D. M. , Sutter, E. G. , Belkoff, S. M. , and Boden, B. P. , 2012, “The Role of Axial Compressive and Quadriceps Forces in Noncontact Anterior Cruciate Ligament Injury: A Cadaveric Study,” American Journal Sports Med., 40(3), pp. 568–573.
13. Xie, F, Yang, L. , Guo, L. , Wang, Z. J. , and Dai, G. , 2009, “A Study on Construction Three-Dimensional Nonlinear Finite Element Model and Stress Distribution Analysis of Anterior Cruciate Ligament,” ASME Journal of Biomechanical Eng., 131(12), p. 121007.