Palestine Polytechnic University

College of Engineering

Femur Bone Model

By

Mostafa Moqadi

Moath Berghith

Mohammad Zalloum

Supervisor:

Dr. Ramzi Qawasma

Submitted to the College of Engineering
In partial fulfillment of the requirements for the
Bachelor degree in Mechatronics Engineering

Hebron, December 2016

Palestine Polytechnic University

Collage of Engineering

Mechanical Engineering Department

Hebron – Palestine

Femur Bone Model

Project Team:

Mostafa Moqadi Moath Bregith

Mohammad Zalloum

Submitted to the Collage of Engineering

In partial fulfillment of the requirements for the

Bachelor degree in Mechatronics Engineering.

Supervisor Signature

……………………………………………

Testing Committee Signature

…………………………………… ……………………………………

Chair of the Department Signature

……………………………………………

May 2017

الإهداء

إلى من جرع الكأس فارغاً ليسقيني قطرة حب

إلى من كلّت أنامله ليقدم لنا لحظة سعادة

إلى من حصد الأشواك عن دربي ليمهد لي طريق العلم والنجاح

إلى القلب الكبير **والدي العزيز**

إلى شمس نهاري وقمر ليلي

إلى نور عيني ومهجة قلبي

إلى من أرضعتني الحب والحنان

إلى رمز الحب وبلسم الشفاء

إلى القلب الناصع بالبياض **والدتي الحبيبة**

إلى القلوب الطاهرة الرقيقة والنفوس البريئة

إلى رياحين حياتي

إلى من يحملون في عيونهم ذكريات طفولتي وشبابي **إخوتي وأخواتي**

إلى الأرواح التي سكنت روحي

الآن تفتح الأشرعة وترفع المرساة لتنطلق السفينة في عرض بحر واسع مظلم هو بحر الحياة

وفي هذه الظلمة لا يضيء إلا قنديل الذكريات ذكريات الأخوة البعيدة

إلى الذين أحببتهم وأحبوني **أصدقائي**

إلى من سرنا سويا ونحن نشق الطريق معا نحو النجاح والإبداع **زملائي وزميلاتي**

إلى من ضحوا بحريتهم من أجل حرية غيرهم **اسرانا البواسل**

إلى من هم أكرم منا مكانة **شهداء فلسطين**

إلى هذا الصرح العلمي الجبار **جامعة بوليتكنك فلسطين**

إلى من احتضنتني كل هذا الكم من السنين **فلسطين الحبيبة**

شكر وعرفان

لله الحمد كله والشكر كله أن وفقنا وألهمنا الصبر على المشاق التي واجهتنا لإنجاز هذا العمل المتواضع.

لابد لنا ونحن نخطو خطواتنا الأخيرة في الحياة الجامعية من وقفة نعود إلى أعوام قضيناها في رحاب الجامعة مع أساتذتنا الكرام الذين قدموا لنا الكثير باذلين بذلك جهودا كبيرة في بناء جيل الغد لتبعث الأمة من جديد...

وقبل أن نمضي نتقدم بأسمى آيات الشكر والامتنان والتقدير والمحبة إلى الذين حملوا أقدس رسالة في الحياة...

"كن عالما .. فإن لم تستطع فكن متعلما ، فإن لم تستطع فأحب العلماء ، فإن لم تستطع فلا تبغضهم".

إلى الذين مهدوا لنا طريق العلم والمعرفة...

إلى جميع أساتذتنا الأفاضل...

ونخص بالتقدير والشكر:

مشرف المشروع الدكتور : **رمزي القواسمة** على ما قدمه لنا طوال فترة العمل وتذليل الصعاب أمامنا

وكذلك نشكر كل من ساعد على إتمام هذا البحث وقدم لنا العون ومد لنا يد المساعدة وزودنا بالمعلومات لإتمام هذا البحث ونخص بالذكر **م. مجدي زلوم**

Abstract

Reverse Engineering in terms of mechanical engineering is the processes of extracting knowledge or design information from anything and re-producing it based on the extracted information. The process usually includes a 3D scanning of the object, transfer the obtained data to the computer for measurements and data sensing based on 3D high quality analysis. The created model using CAD is used in analyzing, modifying, optimizing and manufacturing the desired object or product in factories. This project is aimed to use this technique for studying, analyzing and designing a femoral human bone with the purpose of improving implantable bone properties and reducing the time and the cost of manufacturing. This will be achieved by designing and creating a real model of proximal human femoral bone with accurate geometry and material properties retrieved from 3D scan data. The bone model will be used to do a real investigations on the mechanical behavior of bone structures. The femur bone is analyzed using ANSYS Software under load conditions. Hence the mechanical analysis with heterogeneous material property of bone is varying with individuals. The results of this analysis are helpful for orthopedic surgeons for clinical interest.

الملخص

الهندسة العكسية من منظور الهندسة الميكانيكية تهتم في استخراج المعرفة أوتصميم المعلومات من أي نظام موجود بهدف إعادة تصميمه وإعادة إنتاجه لاحقاً إستناداً إلى المعلومات المستخرجة. وعادةً تتضمن هذه العملية القيام بمسح ضوئي ثلاثي الأبعاد للجسم المراد إعادة إنتاجه، ومن ثم يتم نقل البيانات التي تم الحصول عليها الى جهاز الحاسوب لعمل القياسات الخاصة بالنظام من خلال تحليل ثلاثي الأبعاد عالي الدقة عن طريق برمجيات خاصة. يستخدم النموذج الذي تم إنشاؤه باستخدام الحاسوب (CAD) في تحليل وتعديل وتصنيع الجسم المطلوب .

ويهدف هذا المشروع إلى استخدام هذه التقنية (الهندسة العكسية) لدراسة وتحليل وتصميم عظام الفخذ البشرية بغرض تحسين خصائص العظام التي تزرع في الجسم وتقليل الوقت وتكلفة الإنتاج. وسيتم تحقيق ذلك من خلال تصميم وإنشاء نموذج حقيقي لعظم الفخذ البشري في غاية الدقة من حيث خصائص المواد والأبعاد الهندسية التي تم استخراجها من بيانات الماسح الضوئي للعظمة. سيتم استخدام النموذج العظمي الذي تم تصميمه للقيام بتنبؤ حقيقي على السلوك الميكانيكي لبنية العظم بإختلاف طبقاتها وأبعادها. حيث يتم تحليل عظم الفخذ **(Stress Strain Analysis)** باستخدام برنامج **(ANSYS)** من خلال التأثير عليه بأحمال مختلفة. ومن خلال التحليل الميكانيكي لمواد غير متجانسة تتكون منها عظمة الفخذ وذات خصائص خاصة يمكن تصميم عظام لأشخاص من فئات عمرية مختلفة. وتعد نتائج هذا التحليل مفيدة جداً لجراحّي العظام للاستفادة منها في العناية السريرية .

List of Content

List of Figures

Chapter Three

Chapter Five

Chapter Six

List of Tables

Chapter One

Chapter Five

Chapter Six

List of Abbreviations

3D	Three-Dimensional
2D	Two-Dimensional
1D	One-Dimensional
CAx	(CAD, CAE, CAM)
CAD	Computer Aided Design
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing
DOF	Degree Of Freedom
DMU	Digital Mock Up
PC	Personal Computer
HD	High Definition
CMOS	Complementary Metal Oxide Semiconductor
STL	**ST**ereo**L**ithography
OBJ	Object file
VRML	Virtual Reality Modeling Language
MLT	Multi Laser Triangulation
HVAC	Heating Ventilation and Air-Conditioning
BIW	Body In White
ICEM	Integrated Computer Engineering and Manufacturing

NURBS	Non Uniform Rational Basis Spline
DCC	Digital Content Creation
ROI	Return On Investment
FEA	Finite Element Analysis
FEM	Finite Element Method
FDM	Finite Difference Method
Mimics	Materialise Interactive Medical Image Control System

CHAPTER ONE

Introduction

1.1 Validation Techniques and constraints levels

1.2 Optimization of Geometry

1.3 Project idea and Importance

1.4 Software’s used in the project

1.5 Project Objectives

1.6 Literature Review

1.7 Schedule and time table

1.8 Project Budject

Reverse Engineering can be defined as the process of analyzing a physical object to identify and find the relationship between the particles and components of it, and create representations of the object in another form or at a higher quality. Also they can say that it is the process of duplicating an existing component by capturing the physical dimensions components.

1.1 Validation Techniques and constraints levels

The primary goal of reverse engineering is to create high precision models from the object that reflect to geometry, and the intended design behind the geometry. The knowledge of geometric and parametric constraints is not sufficient to optimize the algorithms. Reverse engineering expresses the issue of constrained optimization that produces more accurate models than the knowledge of geometric and parametric constraints.

The original models are available for comparison with the reverse engineered models through successful and powerful developed techniques that will be discussed. They will overcome the error measurements of the original design.

The accuracy in Modeling depends on the properties of sensor that scans the geometry and relative error of this sensor. There are three levels of constraints that are useful in creating the model, they are:

1. Specific primitives narrow the possible shape of the reconstructed model from arbitrary geometry down to a well-defined set of design and manufacturing features.
2. Specific domain pragmatics attempt to capture specific geometric conditions and conformities that are likely to be found based on how a part is designed and manufactured.
3. Functional constraints describe interaction among the features of the object.

1.2 Optimization of Geometry

Optimization method can be interpreted as a process to minimize some undesirable criteria, it is divided into two cases, constraint and unconstraint optimization.

In the unconstraint case, the optimization occurs just on geometric dimensions between algorithms and sense data points, while in the constrained case, the hypothesized model is created using a limited set of appropriate geometric forms that are then optimized based on the data. It's very important to know geometric constraint because it helps us to prevent the sense data, which leads us to the primitive object.

To achieve good optimization, the constraint and the model should be displayed mathematically. By using the degree of freedom (DOF) which purpose is to reduce the dimensions of the model to be redefined using fewer variables, it is possible to define error metrics for the sensed data, and for the violation of constraints.

1.3 Project idea and Importance

1. It has been noticed that the number of the patients who seek replacement of joints and bones operations in the world are increasing, which calls the researchers and the engineers to search for designs and better alternatives to replace the diseased joints and the bones with the same characteristics and specifications.
2. This project is the first of its kind in the Palestine Polytechnic University.
3. Palestinian society is needed for such research because of the large number of injuries as a result of the occupation.
4. Demonstrate that mechanical engineering is not only interested in machines and plants but also interested in the human body and its members.

1.4 Software's used in the project

The reverse engineering process requires software that reconstructs the object as a 3D model. At the beginning, the physical object can be measured using 3D scanning technologies, like a coordinate measuring machine, laser scanner, structured light digitizer or computed tomography, then a high performance programs require to deal with 3D scanning, and that will be by using a unique software's.

1.4.1 Geomagic Studio

Geomagic Studio it's also an integrated program that transforms the 3D scanned data into highly accurate surface and 3D CAD model, in general it has a professional characteristic that can edit point cloud, making mesh analysis, also it has tools that can interstate the internal structure of any object, which help to create a high quality of 3D models, and Optimize for fast data processing [1].

1.4.2 CATIA

The subject of rigid body dynamics is an integral part of mechanics. In general, dynamics has two distinct branches, kinematics, and kinetics.

The topic of kinematics deals with the motion of objects disregarding the forces that cause the motion. Here, one is interested in studying position, velocity and acceleration On the other hand, kinetics deals with the forces causing motion and resulting an acceleration.

CATIA's Digital Mock Up workbench has a module known as DMU Kinematics where the former type of dynamic simulations can be performed. However, kinetic analysis is not available in CATIA V5. There are add-on third party packages that can incorporate force calculations in the mechanisms designed in CATIA.

To effectively model mechanisms in DMU Kinematics, one should have a thorough understanding of the various kinematic joint types and their associated effect on system degrees of freedom.

The idea of the project is based on studying the vital and mechanical ingredients, components and characteristics of the Femur in terms of durability and locating its strength and weakness for this important bone in the human body. The bone will be scanned by 3D scanner and the image will be used to build 3D model using special software to simulate the Femur which have the same vital and mechanical characteristics for the real bones [2].

1.4.3 ANSYS

ANSYS is a general purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers, which enables to simulate tests or working conditions, enables to test in virtual environment before manufacturing prototypes of products. Furthermore, determining and improving weak points, computing life and foreseeing probable problems are possible by 3D simulations in virtual environment, and gives an opportunity for taking needed features. ANSYS can work integrated with other used engineering software on desktop by adding CAD and FEA connection modules.

ANSYS Mechanical is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. This computer simulation product provides finite elements to model behavior, and supports material models and equation solvers for a wide range of mechanical design problems. ANSYS Mechanical also includes thermal analysis and coupled-physics capabilities involving acoustics, piezoelectric, thermal–structural and thermo-electric analysis.

Through Finite element analysis means simulating nature, therefore giving you insight into the world you live in and enriching your life.

Start by modeling a lifting draw component and solve first 3D finite element model. Then, creating first 3D model, and setting up the loads and boundary conditions. Finally, view the results and ensure the safety of the component under the given loads. After watching this course, to develop and analyze your own models [3].

1.5 Project Objectives

1. Study of biomechanical properties of human bones.
2. Create a model that simulates human femur bones.
3. Determining the strength and weakness points of the femur bone.
4. Analyze the forces and loads that affect the femur bone using ANSYS software.

1.6 Literature Review

"Biomechanical Analysis of Human Femur Bone"
India, 2011
Raji Nareliya, Veerendra Kumar

Biomechanics is the theory of how tissues, cells, muscles, bones, organs and the motion of them and how their form and function are regulated by basic mechanical properties. A finite element model of bones with accurate geometry and material properties retrieved from CT scan data are being widely used to make realistic investigations on the mechanical behavior of bone structures. The aim of this study is to create a model of real proximal human femur bone for evaluating the finite element analysis (FEA). Here, behavior of femur bone is analyzed in ANSYS under physiological load conditions. Hence the mechanical analysis with heterogeneous material property of bone is varying with individual patient. The results of this analysis are helpful for orthopedic surgeons for clinical interest [4].

"Biomechanical Investigation of Human Femur by Reverse Engineering as a Robust Method And Applied Simplifications"

Iran, 2013

A. Latif Aghili, A.M. Goudarzi, M.H. Hojjati, S.M. Rabiee, Misagh Imani and Ali Paknahad

Stress fracture is a type of biomechanical failure of bone caused by loads during intense physical training. This failure is very important for aged persons and athletics. Because of cell death, in some cases after emergency surgery the injured person may suffer from lifelong disability. In this research, our model was construct by means of reverse engineering method and this three-dimensional finite element model of human femur has been analyzed under single, expanded and partial expanded loads. Analysis has been performed using commercially available software. The material is assumed to have isotropic elastic characteristics. The results indicated that the important stress occurred at the inferior root of the femoral neck but maximum stress obtained at the femoral shaft. The magnitude of the strain shows good agreement with the published experimental results. This verifies the finite element modeling and the simplified model used [5].

"Stress Fracture Risk Analysis of the Human Femur Based on Computational Biomechanics"

USA, 2004

Liming Voo, Mehran Armand, and Michael Kleinberger

The lack of predictive tools to assess the relative risk of stress fracture injury has posed a great challenge for military readiness and competitive athletic training. This article presents a methodology of assessing such risks using computational models and biomechanical stress analysis in a human femur. Preliminary results have demonstrated the capability of our methodology in studying the effect of risk factors as independent variables, which would be very difficult, if not impossible, to accomplish by any other means. We have shown that certain geometric features are significant risk factors for femoral neck stress fracture. Further development of our methodology will enable more accurate prediction of stress fracture risk and will help in the design of optimal training regimens to minimize the risk of injury [6].

"Research Using Finite Element Method of Biomechanical Behaviours of Human Femur Model Under The Different Loads"

Turkey, 2014

Kadir Gok*, Betul Taspinar, Ferruh Taspinar, Emrah Afsar, Arif Gok, M. Burak Bilgin
In this study, analysed using finite element method (FEM) of biomechanical behaviours of human femur model under the different loads. Geomagic and Soldiworks program were used for 3D model human of femur model, and Ansysworkbench was used for finite element analysis (FEA). Compression, tension, bending and torsional loading were applied to 3D human femur model. Femur bone was exhibited that the best stabile behavior under the torsional loading. The reason is due to the bone structure. In contrary to this situation, the femur model under the bending loading type has lowest safety factor due to high stress and deformation. Therefore, type of loading on the femur is crucial for surgical procedure and rehabilitation approaches [7].

1.7 Schedule and time table

Table 1-1: First semester time plan

Task \ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Project selection	■	■													
Collecting from different information			■												
Introduction Chapter 1			■	■	■										
Chapter 2						■	■								
Chapter 3						■	■	■	■	■					
Chapter 4										■	■	■			
Chapter 5											■	■	■	■	■

Table 1-2: second semester time plan

Task \ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Get Cow Femur	■	■	■												
3D Scanning				■											
Geomagic Software					■	■	■								
ANSYS Software								■	■	■	■	■	■		
3D Printer														■	■
Chapter 6			■			■	■	■	■	■	■	■	■	■	■

1.8 Project Budject

Table 1-3: Total cost of the project

Components	Pries (JD)	Quantity	Total (JD)
3D scanner of femur bone	200	3	600
3D Printer of femur bone	200	3	600
قرطاسية	100	1	100
			1300

CHAPTER TWO

Theoretical Background

2.1 Reverse Engineering in Factories

2.2 Data Segmentation and Fitting

2.3 Geometric Constraints

2.1 Reverse Engineering in Factories

The main purpose of reverse engineering in manufacturing is repeating the original objects, which exploit whether in the modifying after the initial design stage or in maintenance process (as a spare parts), this process has found use in computer aid designs and animation.

Raband et al. [8] identifies the results of a reverse engineering operation as producing a type three drawing set and a set of intelligent CAD models of the components. Further, they define the reverse engineering preprocess as:

1. Collecting all available information and documentations, including nonproprietary drawings, functional requirements, tooling requirements, processing and material requirements, etc.
2. Identifying new data elements required for a complete technical data package.
3. Performing a cost/benefit analysis.
4. Contacting the cognizant engineer.
5. Establishing a reverse engineering management plan.
6. Establishing acceptance criteria.

When these considerations have been applied, the processes of reverse engineering must be precisely and clear. The stage of collecting information and documentation indicates to scan the object, that is often simply includes a set of 3D geometric points associated with the surface of the object. Broacha and Young [9]. They discuss the important points that should be provided in any reverse engineering system:

1. These include an ability to collect data.
2. A geometrical foundation of surface modeling which depends on the data.
3. Comprehensive functionality for displaying and manipulating point data.
4. The actual process for reverse engineering or surfacing.

These established reverse engineering systems should be flexible and easy to those who are interested in this matter.

2.2 Data Segmentation and Fitting

The goal of data segmentation is to associate data points with the hypothesized feature they represent. Proper classification of data points into their associated features is necessary before fitting can begin. This process starts from the vision community during it the vision images partitioning on the main benefit. There are two primary methods exist for segmentation of a range image, first is the edge based. It’s a method use discontinuities to encircle a region that is then considered classified. The second is the region based segmentation, it’s a method try to classify points based on local properties, such as intensity value, orientation, or curvature [10].

3D point cloud segmentation is the process of classifying point clouds into multiple homogeneous regions, the points in the same region will have the same properties [11].

3D segmentation of clouds depends on the two approaches, which are the bottom-up segmentation, and top-down segmentation. Recently the work suggests to work top-down segmentation because the bottom-up segmentation lines, planes, and arcs, are identified and then combined into shapes such as pockets or outlines. This technique can fail when the data is extremely noisy, or when the surface does not conform to standard assumptions of smoothness and local uniformity.

Data fitting techniques in general include approximation and interpolation. In the simplest case, interpolation techniques fit functions directly through the measured data points. Approximation techniques fit functions in the neighborhood of the data points, attempting to minimize some error function. Interpolation is often used in the design process where the designer represents a shape, such as the profile of the part, by several points and asks the computer to connect them via primitives such as arcs, lines, or splines [12].

Lots of CAD programs provided with spline based model [10], that is because splines supply a mathematically, a good representation for 3D curves and surfaces that which has a superb feature such as smoothness constraints and data reduction. Splines can also be broken up into local sections to model more complex geometry. These local sections can be modified without affecting any other part of the surface. Unfortunately, splines are a generic approximation of a surface they can undulate through the noise reducing the error to the data, so splines can actually over fit the data. This can produce a curved surface where a lower DOF surface, such as a line or arc, is more appropriate.

2.3 Geometric Constraints

Geometric constraints describe the physical representation of the geometry, in order to create a mathematical model, describe the features of the object.

Many strategies have been tried, beginning as early as the 1960s with Sutherland's Sketchpad program. Hsu's dissertation [13], attempts to address the idea of geometric constraint solving in the design process. He lists several criteria for an ideal constraint. He lists several criteria for an ideal constraint solver:

1. Reliability - Derive all possible solutions (if required).
2. Predictability - Do not jump erratically through the solution space and should provide a way for a human to control the results.
3. Efficiency - Allow interactive response times.
4. Robustness - Handle over and under-constrained problems.
5. Generality - Handle a wide variety of constraint types and not be restricted to any Specific dimensions.

Couching these requirements in terms of a computer aided reverse engineering system gives:

1. Reliability - The algorithm should derive a model that is consistent with the data given it and its knowledge of the design and manufacturing processes.
2. Predictability - The algorithm should come up with the simplest accurate solution. An interactive user should be able to guide the process.
3. Efficiency - The algorithm should run at interactive speeds.
4. Robustness - The algorithm should be able to handle over and under-constrained hypothesized features.

5. Generality - The algorithm should be able to handle a wide variety of parts and inter-related constraints and not be restricted to any specific dimensions.

Through these assumption, Hsu defines four methods developed to address the constraint-solving problem. These include propagation methods that is the process of representing the geometrical constraints in the form of an acyclic graph. Numerical methods, that described previously in relation to data fitting, and geometry is represented as algebraic formulas and constraints are created by relating variables across equations. Constructive methods unfortunately, these techniques are sometimes unpredictable and can have difficulties converging, so there is a newer methods of constraint solving which applies to problems solvable by ruler-and-compass construction is known as the constructive approach.

In the rule-based approach, geometric constraints are represented symbolically. Rewrite rules are utilized to simplify geometry and reduce DOFs. Unfortunately, rule-based systems tend to be slow [12]. Graph-based approaches consist of two steps. One, a top-down phase is entered where the graph is analyzed and a sequence of constructive steps is derived. Two, a bottom-up phase occurs where the construction steps are carried out and the model is constructed.

The final method that Hsu addressed is algebraic methods, which the geometric constraints are written as algebraic formulas, which are then combined and reduced using elimination methods. This method tends to be extremely slow and often have exponential complexity.

CHAPTER THREE

Reverse Engineering Processes

3.1 Scanning by using 3D scanner

3.2 Benefits of Project Software

3.3 Mesh Preparation

3.1 Scanning by using 3D scanner

3D scanning is getting popular in various fields, and its usage for different purpose, this means that there are a different kinds objects, simple or complex can collect the data of them, and many details to understand.

Collecting data about object in two-way hand held and modern scanning. Traditional scanning or manually driven sensors such as calipers and micrometers. These devices fall within the paradigm of touch sensing, requiring some sort of probe to physically contact the surface of the part, within using a modern sensing tools, other is modern scanning within use 3D scanners.

3D scanners are a technology that digitally captures the shape of physical objects using a line of laser light. 3D scanners create “point clouds” of data from the surface of an object. In other words, 3D scanning is a way to capture a physical object’s exact size and shape into the computer world as a digital 3D representation, with high resolution and minimum error data.

3.1.1 Types of scanning technologies

In these days, there are many types of 3D scanning technologies on the market today. The most commonly used technologies fall into three categories, Displacement, Profile, and Snapshot (aka, Scanner) [14].

- Displacement devices use a single point laser beam projection as in Figure 3.1 to measure the height, thickness, or position of a project.

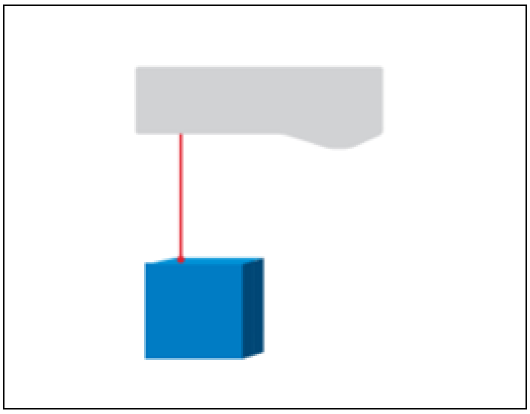


Figure 3-1: Displacement devices

- Line Profile devices typically use a projected laser line to create a cross action profile for measuring aspects of an object's contour. Moving an object under the laser line creates many profiles that can be combined into a complete 3D shape. Figure 3.2 describe the process of type.

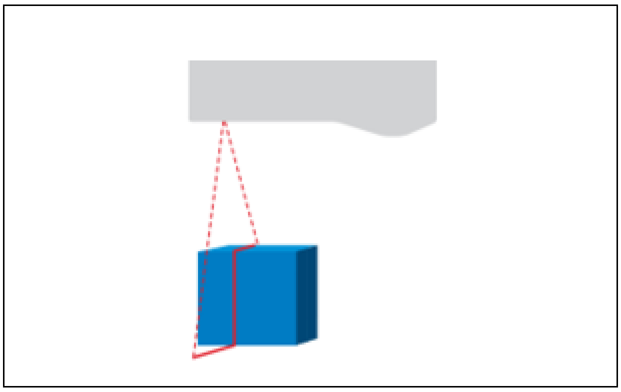


Figure 3-2: Line Profile devices typically

- Snapshot devices as in Figure 3-3, use structured light (non-laser) and stereo-vision to generate full 3D volume data. Because Snapshot technology captures so much 3D data at one time, objects need to remain stationary during the scanning process.

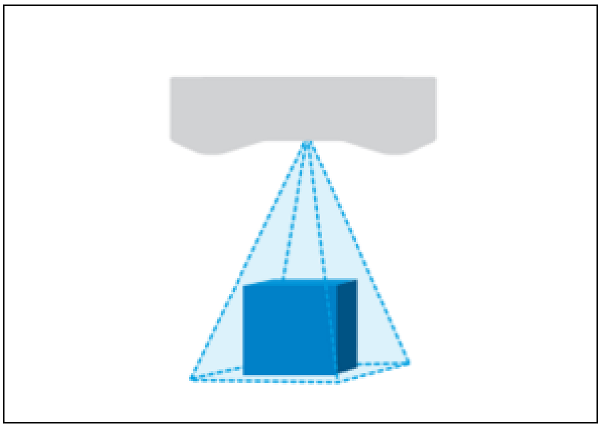


Figure 3-3: Snapshot devices

3.1.2 Power of 3D scanning data

A 3D scanner is a device that captures a real object or environment as 3D shape data. The collected 3D information is converted into digital data commonly called 3D scan data or scan data. 3D scan data is a set of points. A point represents a location on a real object and contains the X, Y, and Z coordinates. Numerous points can be used to describe a real object. For example, a digital photo with a high resolution pixel count can represent the detailed shape of a real object. Figure 3-4.

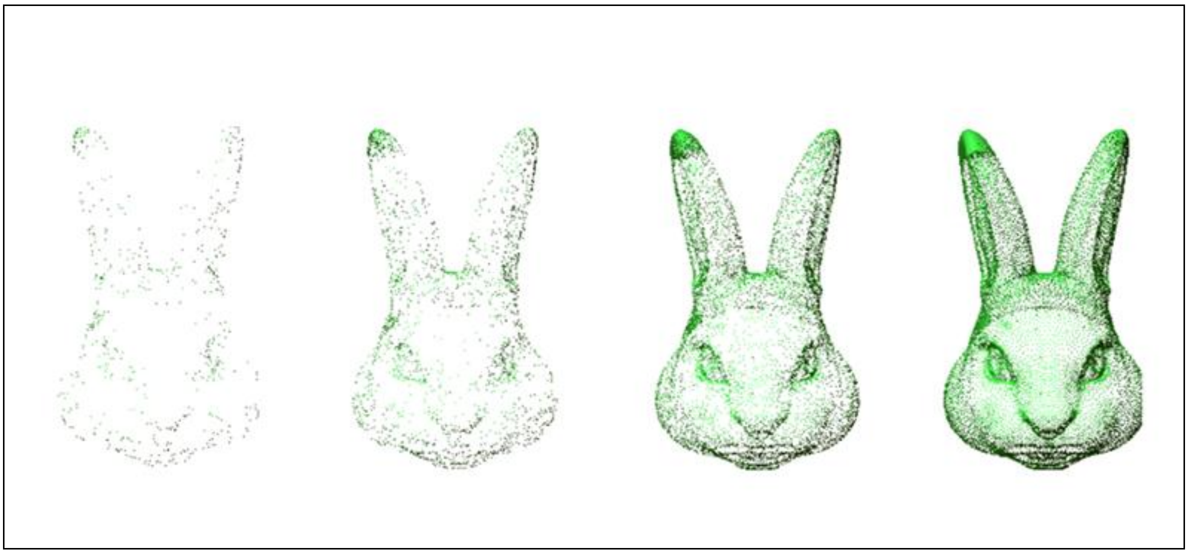


Figure 3-4: 3D scan data by resolution

A point set, also known as a point cloud, can be converted into an informative digital model with software operations and used in various industrial fields. 3D scan data has the following strengths.

- Quickly create digital versions of real objects
- Accuracy
- Capture complex freeform surfaces
- Capture small to large scale objects (depending on 3D scanner)
- Obtain color information (depending on 3D scanner)
- Simulate environments and situations
- Change to different scales and measurements easily
- Easily extract length, height, width, volume and position data
- Extract sectional information
- Compatibility in a general PC environment

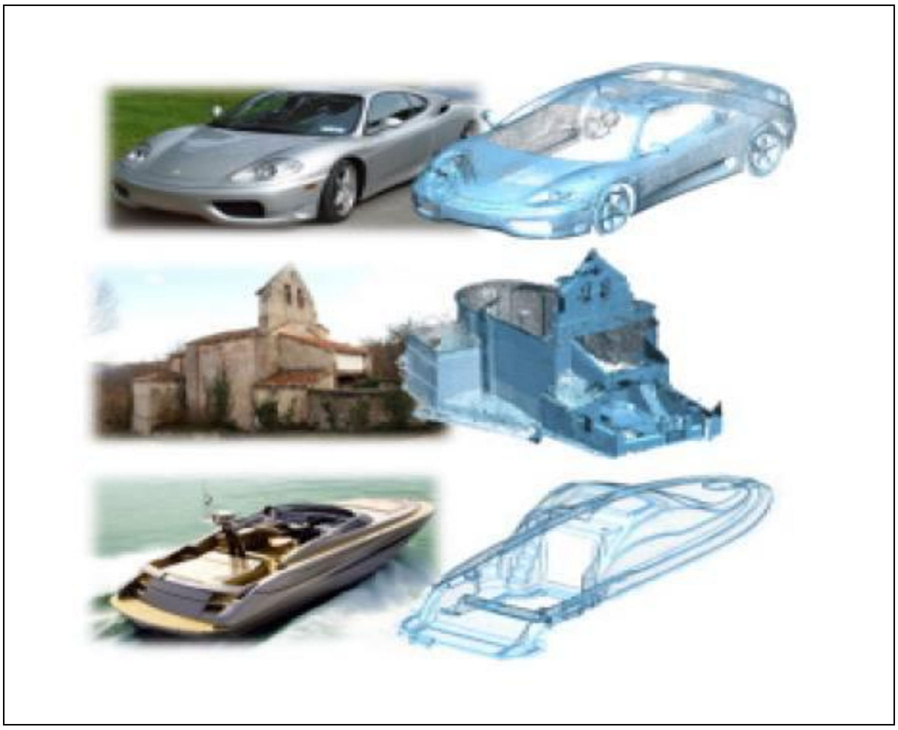


Figure 3-5: 3D scan data

Since 3D scan data can represent a real object with high accuracy, it is used for various purposes. The use of scan data is also increasingly expanding to custom markets as scan technology becomes easier to use and more intuitive. Figure 3-5 show scan data.

3.1.3 Measurement and Inspection

3D scan data is increasingly used for metrology or inspection. The more accurate points that are used to measure a feature will increase the credibility of an inspection. The scan data of a manufactured part can be aligned with its CAD model (nominal data), and subsequently compared to check for differences and whether or not they pass/fail within set tolerances.

3.1.4 Ultra HD 3D scanner

This is one of scanners, which has a superb feature such as:

- The dimensional accuracy of this it is about ±100 micron in micro mode and ±300 micron in wide mode.
- The process speed about 50 thousand points per second.
- Twin 5 megapixel CMOS image sensor.
- Scans can be output as STL, OBJ, VRML, XYZ, and PLY files, which is available with geomagic back ages.

This type use Multi Stripe Laser Triangulation (MLT) Technology, this technology is relatively cheaper than others, so it can be available for schools, universities and some people interested with reverse engineering, in Figure 3-6 shows the components of this scanners.

Figure 3-6: Multi Stripe Laser Triangulation (MLT)

3.2 Benefits of Project Software

3.2.1 Benefits of CATIA

CATIA an acronym of (computer aided three-dimensional interactive application) Commonly referred to as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAX), including conceptualization, design (CAD), engineering (CAE) and manufacturing (CAM). CATIA facilitates collaborative engineering across disciplines around its 3DEXPERIENCE platform, including surfacing & shape design, electrical fluid & electronics systems design, mechanical engineering and systems engineering. CATIA facilitates the design of electronic, electrical, and distributed systems such as fluid and HVAC systems, all the way to the production of documentation for manufacturing.

CATIA enables the creation of 3D parts, from 3D sketches, sheet metal, composites, molded, forged or tooling parts up to the definition of mechanical assemblies. The software provides advanced technologies for mechanical surfacing & BIW. It provides tools to complete product definition, including functional tolerances as well as kinematics definition. CATIA provides a wide range of applications for tooling design, for both generic tooling and mold & die. it also, offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA supports multiple stages of product design whether started from scratch or from 2D sketches [2].

3.2.2 Benefits of Geomagic Studio

- Immediately and rapidly create high quality 3D data from almost any source.
- Increase productivity in design, manufacturing and repair workflows by quickly and accurately testing and redesigning existing parts and objects in 3D.
- It can Access the fastest path to cad from point and polygon data.
- Optimize work time by using the fastest path to CAD from point and polygon data.
- Access the industry best tools for reverse engineering.
- Deliver and Design Revolutionary New Products and Treatments
- Deliver results immediately.
- Leverage the physical objects you already have into digital assets.

As any engineering software, Geomagic Studio has a unique feature that make it distinct from others. It’s Available in 9 languages. It’s automated and flexible, it can deliver the best experience and the highest quality 3D data from scan data Integrates accurate, its deal CATIA, Autodesk Inventor, SolidWorks and Cero Elements/Pro parametric CAD systems, by export a suitable file with these programs for mediate use in design and editing. It can export 3D data in all major neutral polygonal and NURBS in high quality formats for immediate downstream use Comprehensive support, technical help, training, and free trials of the Geomagic products [1].

In the newest available version of Geomagic Studio 2012, it has Top New Features such as :

- Self-evident for new Sketch function allows direct creation and editing of cross-section curves from both point clouds and polygon models.

- Powerful scripting environment extends, customizes and automates functions with deep access into selected commands in the software.
- Improved editing, navigation and visualization of point clouds from mid- and long-range scanners for scene level 3D models.
- Remeshing tool enables fast, accurate triangulation of polygon models for cleaner, more usable 3Dmodelsfor Digital Content Creation (DCC) and 3D printing.
- New ‘Patch’ command delivers superior power for quickly and accurately repairing polygon models.

3.2.3 Benefits of ANSYS

NSYS is an easy-to-use virtual-prototyping and modular simulation system that extends to meet your needs, making it a low-risk investment you can expand as value is demonstrated. It is scalable to all your organizational levels, your degree of analysis complexity and your stage of product development.

ANSYS advantages and benefits are well documented. According to studies, best-in-class companies perform more simulations in less time. As a leader in virtual prototyping, ANSYS is unmatched in the functionality and power necessary to optimize components and systems [3].

With ANSYS software, you can achieve:

- Innovative, reliable and high-quality products and processes
- Faster ROI, due to fewer physical prototypes and test setups
- A more flexible and responsive information-based development process, enabling modifications of design at later stages
- A seamless working exchange of data, regardless of location, industry, CAD environment, etc.

3.3 Mesh Preparation

After creating model, for further Finite element analysis (FEA) Figure 3-7,[4] surface mesh is generated for femur bone model. Remeshing is one of the module provided in Mimics for create meshing. In automatic remeshed operation surface mesh of equilateral triangle is generated. All parameters must be optimized here with the ratio of the height of the triangle and the length of its base and resulting triangles possesses quality parameter value above the maximum threshold 0.3. In remeshing following actions were performed:

- Amount of details are reduced.
- Amount of triangles of the model is reduced.
- Qualities of the triangles are improved.
- The amount of triangles while preserving the quality is reduced.
- Extra shells are removed.
- Intersecting triangles are eliminated completely.

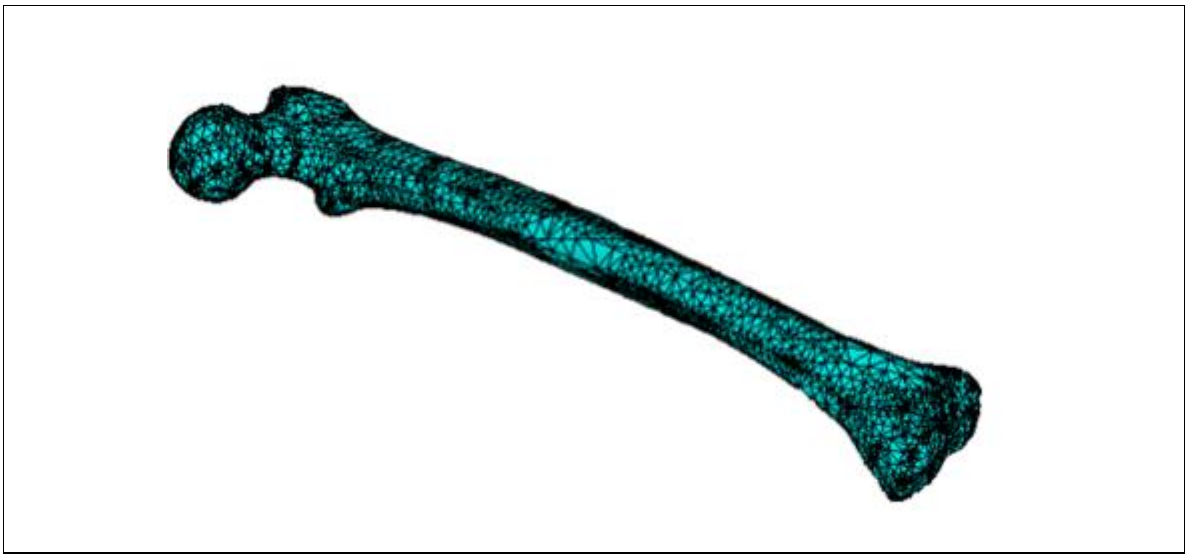


Figure 3-7: FEA Mesh of Femur Bone

This surface mesh can be used to generate a volumetric mesh in FEA preprocessor. The volumetric mesh can be generated in ANSYS for the model of femur bone. The FEA software

Geomagic Studio was used for generating volumetric mesh, which consume less time in comparison to ANSYS. The surface mesh of femur is imported in Geomagic Studio, and converts this surface mesh into volumetric mesh by using conversion option in mesh edit tri to tetra. The three dimensional model of femur bone with volumetric mesh is used for FEA analysis. Figure 3-8(a) shows surface mesh of femur bone and Figure 3-8(b) shows volumetric mesh of femur bone in mimics.

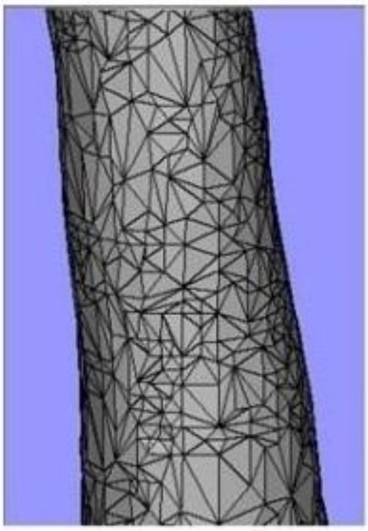


Figure 3-8(a): Surface Mesh

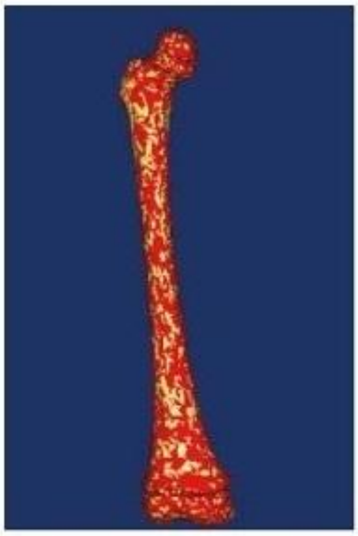


Figure 3-8(b): Volumetric Mesh of femur bone

CHAPTER FUOR

Finite Element Method (FEM)

4.1 Introduction and definition

4.2 Basic Steps

4.3 The Purpose of FEA

4.4 Discretization

4.1 Introduction and definition

A finite element method (abbreviated as FEM) is a numerical technique to obtain an approximate solution to a class of problems governed by elliptic partial differential equations. Such problems are called as boundary value problems as they consist of a partial differential equation and the boundary conditions. The finite element method converts the elliptic partial differential equation into a set of algebraic equations which are easy to solve. The initial value problems which consist of a parabolic or hyperbolic differential equation and the initial conditions (besides the boundary conditions) cannot be completely solved by the finite element method.

The parabolic or hyperbolic differential equations contain the time as one of the independent variables. To convert the time or temporal derivatives into algebraic expressions, another numerical technique like the finite difference method (FDM) is required. Thus, to solve an initial value problem, one needs both the finite element method as well as the finite difference method where the spatial derivatives are converted into algebraic expressions by FEM and the temporal derivatives are converted into algebraic equations by FDM. Useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained.

3D finite element analysis is the only available technique that accounts for the complexity of the hip geometry and its material the complete human femur 3D-model imported into the finite element commercial software and because of irregular surface of femur 3D-model, it was meshed by one element with middle node. This element has a quadratic displacement and is well suited to model irregular meshes (such as those produced from various CAD System) [15].

4.2 Basic Steps

The finite element method involves the following steps:

- First, the governing differential equation of the problem is converted into an integral form. These are two techniques to achieve this: (i) Variational technique and (ii) Weighted residual technique. In variation technique, the calculus of variation is used to obtain the integral form corresponding to the given differential equation. This integral needs to be minimized to obtain the solution of the problem. For structural mechanics problems, the integral form turns out to be the expression for the total potential energy of the structure. In weighted residual technique, the integral form is constructed as a weighted integral of the governing differential equation where the weight functions are known and arbitrary except that they satisfy certain boundary conditions. To reduce the continuity requirement of the solution, this integral form is often modified using the divergence theorem. This integral form is set to zero to obtain the solution of the problem. For structural mechanics problems, if the weight function is considered as the virtual displacement, then the integral form becomes the expression of the virtual work of the structure [7].

- In the second step, the domain of the problem is divided into a number of parts, called as elements. For one-dimensional (1-D) problems, the elements are nothing but line segments having only length and no shape. For problems of higher dimensions, the elements have both the shape and size. For two-dimensional (2D) or axi-symmetric problems, the elements used are triangles, rectangles and quadrilateral having straight or curved boundaries. Curved sided elements are good choice when the domain boundary is curved. For three-dimensional (3-D) problems, the shapes used are tetrahedron and parallelepiped having straight or curved surfaces. Division of the domain into elements is called a mesh.

- In three step, over a typical element, a suitable approximation is chosen for the primary variable of the problem using interpolation functions (also called as shape functions) and the unknown values of the primary variable at some pre-selected points of the element, called as the nodes. Usually polynomials are chosen as the shape functions. For 1-D elements, there are at least 2 nodes placed at the end-points. Additional nodes are placed in the interior of the element. For 2-D and 3-D elements, the nodes are placed at the vertices (minimum 3 nodes for triangles, minimum 4 nodes for rectangles, quadrilaterals and tetrahedral and minimum 8 nodes for parallelepiped shaped elements). Additional nodes are placed either on the boundaries or in the interior. The values of the primary variable at the nodes are called as the degrees of freedom.

To get the exact solution, the expression for the primary variable must contain a complete set of polynomials (i.e., infinite terms) or if it contains only the finite number of terms, then the number of elements must be infinite. In either case, it results into an infinite set of algebraic equations. To make the problem tractable, only a finite number of elements and an expression with only finite number of terms are used. Then, we get only an approximate solution. (Therefore, the expression for the primary variable chosen to obtain an approximate solution is called an approximation). The accuracy of the approximate solution, however, can be improved either by increasing the number of terms in the approximation or the number of elements [16].

- In the fourth step, the approximation for the primary variable is substituted into the integral form. If the integral form is of variation type, it is minimized to get the algebraic equations for the unknown nodal values of the primary variable. If the integral form is of the weighted residual type, it is set to zero to obtain the algebraic equations. In each case, the algebraic equations are obtained element wise first (called as the element equations) and then they are assembled over all the elements to obtain the algebraic equations for the whole domain (called as the global equations). In this step, the algebraic equations are modified to take care of the boundary conditions on the primary variable. The modified algebraic equations are solved to find the nodal values of the primary variable.

- In the last step, the post-processing of the solution is done. That is, first the secondary variables of the problem are calculated from the solution. Then, the nodal values of the primary and secondary variables are used to construct their graphical variation over the domain either in the form of graphs (for 1-D problems) or 2-D/3-D contours as the case may be [15][16].

4.3 The Purpose of FEA

4.3.1 Analytical Solution

- Stress analysis for trusses, beams, [7][15]. and other simple structures are carried out based on dramatic simplification and idealization:
    1. mass concentrated at the center of gravity.
    2. beam simplified as a line segment (same cross-section).

- Design is based on the calculation results of the idealized structure & a large safety factor (1.5-3) given by experience.

4.3.2 FEM

- Design geometry is a lot more complex; and the accuracy requirement is a lot higher. We need

1. To understand the physical behaviors of a complex object (strength, heat transfer capability, fluid flow, etc…).
2. To predict the performance and behavior of the design; to calculate the safety margin; and to identify the weakness of the design accurately.
3. To identify the optimal design with confidence.

4.3.3 Common FEA Applications

- Mechanical/Aerospace/Civil/Automotive Engineering
- Structural/Stress Analysis
  1. Static/Dynamic
  2. Linear/Nonlinear
- Fluid Flow
- Heat Transfer
- Electromagnetic Fields
- Soil Mechanics
- Acoustics
- Biomechanics

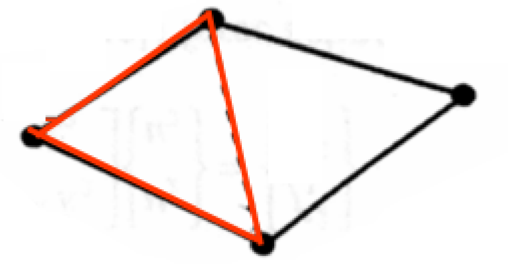
4.4 Discretization

Model body by dividing it into an equivalent system of many smaller bodies or units (finite elements) interconnected at points common to two or more elements (nodes or nodal points) and/or boundary lines and/or surfaces [15].

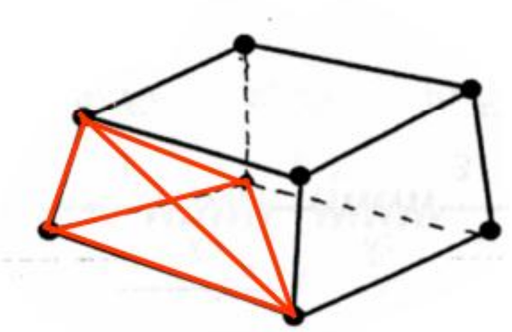
4.4.1 Type of FEM

1- Line element:

2- Plane element:



3- Solid element:



4.4.2 Advantages of the finite element method over other numerical methods are as follows:

- The method can be used for any irregular-shaped domain and all types of boundary conditions.
- Domains consisting of more than one material can be easily analyzed.
- Accuracy of the solution can be improved either by proper refinement of the mesh or by choosing approximation of higher degree polynomials.
- The algebraic equations can be easily generated and solved on a computer. In fact, a general purpose code can be developed for the analysis of a large class of problems [16].

CHAPTER FIVE

Human Bone Modeling

5.1 sequence of the human bone model process

5.2 Parts of the Femur Bone

5.3 Mechanical Properties of Bone

5.4 Loads Applied to Bone

5.1 sequence of the human bone model process

As a start, 3D scan will be made on a Femur and then making a design for the bone using special software (Geomagic Studio, CATIA, ANSYS). Geomagic Studio is used to process the image which has been gained from 3D scanner and then by using CATIA, a model will be made for the bone with the same dimensions and measurements in the image which been processed using Geomagic Studio. By using ANSYS, the model characteristics is been examined by exposing the model to external loads to examine the durability with the possibility to change the magnitude of the load and the location of the impact force. In the case of fracture in one of the parts of the bone model due to the excessive force, it is supported by platinum plates and screws along the bone to increase the durability. After the success of the examination of durability for the model, the Femur model is plotted by 3D printer. In Figure 5-1 show Block Diagram of Project. The idea will be generalized to all of the bones in the human body with the same principle later.

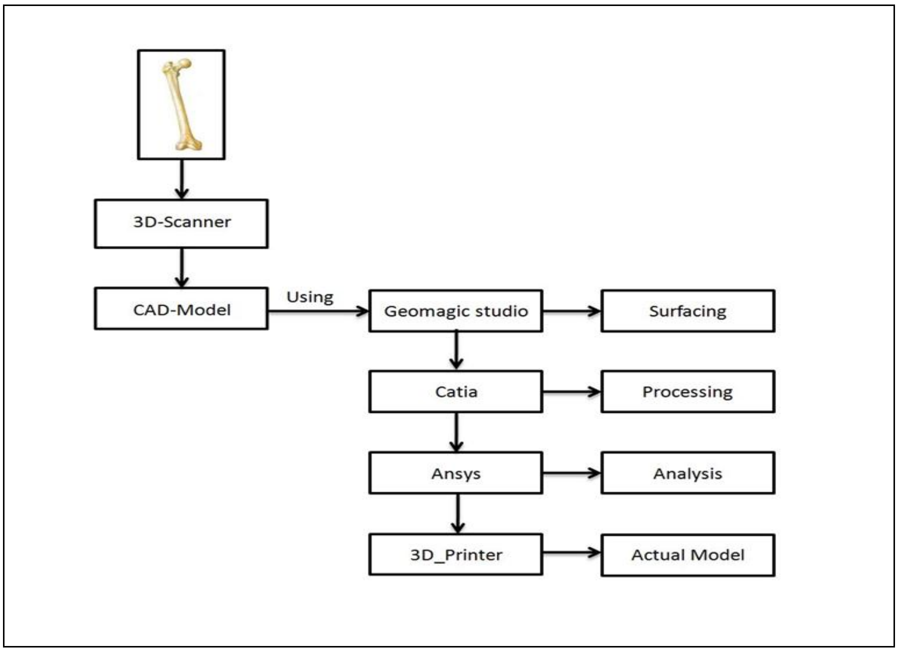


Figure 5-1: Block Diagram of Project

5.2 Parts of the Femur Bone

The proximal area of the femur forms the hip joint with the pelvis. It consists of a head and neck, and two bony processes called trochanters. There are also two bony ridges connecting the two trochanters. In Figure 5-2 show Parts of the femur [17][19].

- **Head –** Articulates with the acetabulum of the pelvis to form the hip joint. It has a smooth surface with a depression on the medial aspect; for the attachment of the ligament of head of femur.

- **Neck –** Connects the head of the femur with the shaft. It is cylindrical, projecting in a superior and medial direction – this angle of projection allows for an increased range of movement at the hip joint.

- **The Shaft**

  The shaft descends in a slight medial direction. This brings the knees closer to the body's center of gravity, increasing stability.

  On the posterior surface of the femoral shaft, there are roughened ridges of bone, these are called the linea aspera (Latin for rough line).

  Proximally, the medial border of the linea aspera becomes the pectineal line. The lateral border becomes the gluteal tuberosity, where the gluteus maximus attaches.

  Distally, the linea aspera widens and forms the floor of the popliteal fossa, the medial and lateral borders form the medial and lateral supracondylar lines. The medial supracondylar line stops at the adductor tubercle, where the adductor magnus attaches.

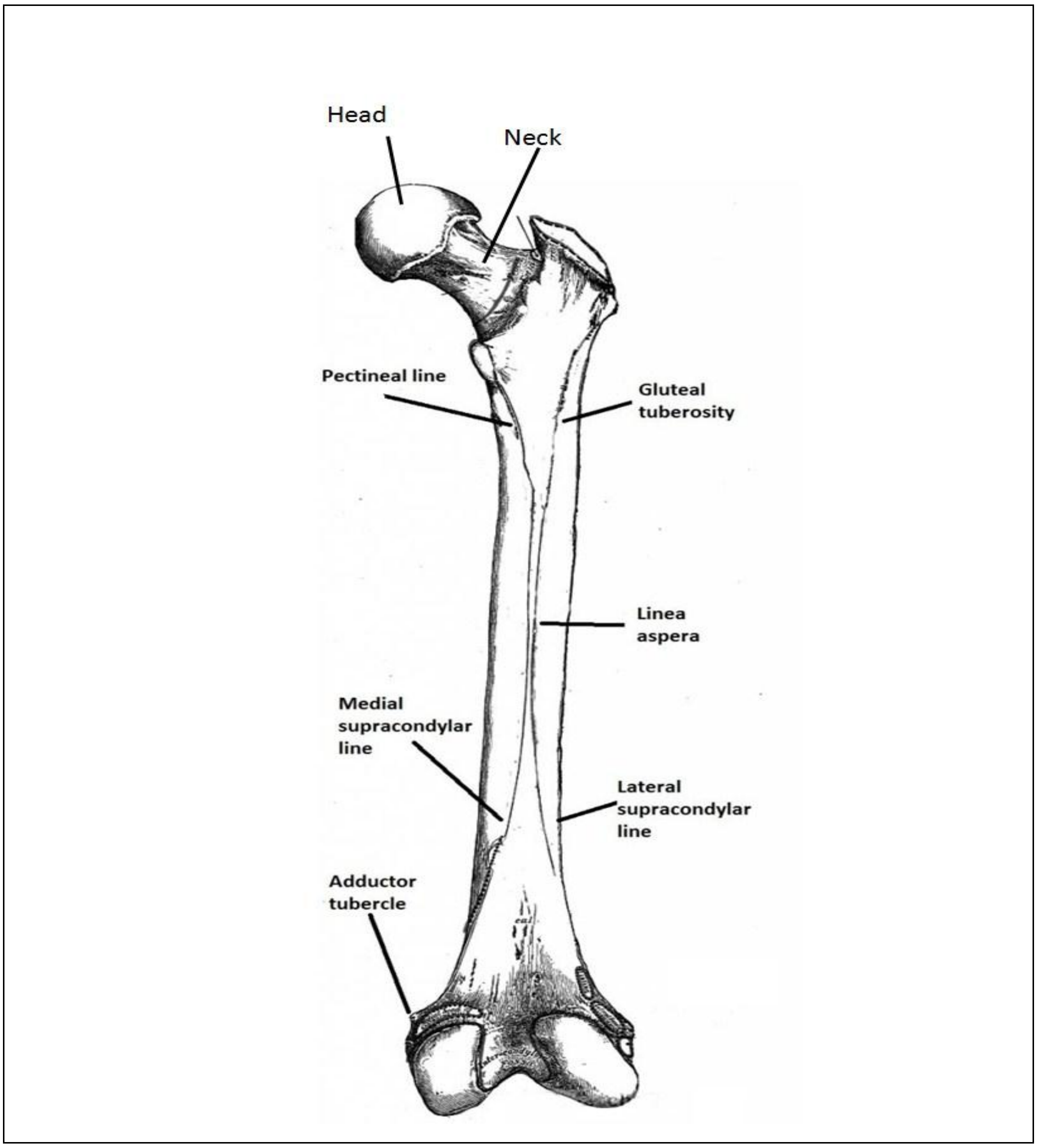


Figure 5-2: Parts of the femur

5.3 Mechanical Properties of Bone

The mechanical properties of bone are as complex and varied as its composition. The measurement of bone strength, stiffness, and energy depends on both the material composition and the structural properties of bone. In addition, the mechanical properties also vary with age and gender and with the location of the bone, such as the humerus versus the tibia. Additional variation may result from other factors such as orientation of load applied to the bone, rate of load application, and type of load.

Bone must be capable of withstanding a variety of imposed forces simultaneously. In a static position, bone resists the force of gravity, supports the weight of the body, and absorbs muscular activity produced to maintain the static posture. In a dynamic mode such as running, the forces are magnified many times and become multidirectional [18].

5.3.1 Strength and Stiffness of Bone

The behavior of any material under loading conditions is determined by its strength and stiffness. When an external force is applied to a bone or any other material, an internal reaction occurs. The strength can be evaluated by examining the relationship between the load imposed (external force) and the amount of deformation (internal reaction) occurring in the material.

As noted earlier, bone must be stiff yet flexible and strong yet light. Strength is necessary for load bearing, and lightness is necessary to allow movement. The strength in weight-bearing bones lies in their ability to resist bending by being stiff. Flexibility is needed to absorb high-impact forces, and the elastic properties of bone allow it to absorb energy by changing shape without failing and then return to its normal length. If the imparted energy exceeds the zone of elastic deformation, plastic deformation occurs at the price of microdamage to the bone. If both the elastic and plastic zones are exceeded, the imparted energy is released in the form of a fracture [18].

5.3.1.1 Strength

The strength of bone or any other material is defined by the failure point or the load sustained before failure. The overall ability of the bone to bear a load depends on having sufficient bone mass with adequate material properties and fiber arrangement that resist loading possibilities in different directions (40). The failure of bone depends on the type of load imposed (Figure 5-3); there is actually no standardized strength value for bone because the measurement is so dependent on the type of bone and testing site. Failure of bone involves either a single traumatic event or the accumulation of microfractures. Thus, both fracture and fatigue behaviors of bone are important. Strength of bone is provided by the mineralization of its tissue: the greater the tissue mineral content, the stiffer and stronger the material. If bone becomes too mineralized, however, it becomes brittle and does not give during impact loading. Strength is assessed in terms of energy storage or the area under a stress–strain curve [18].

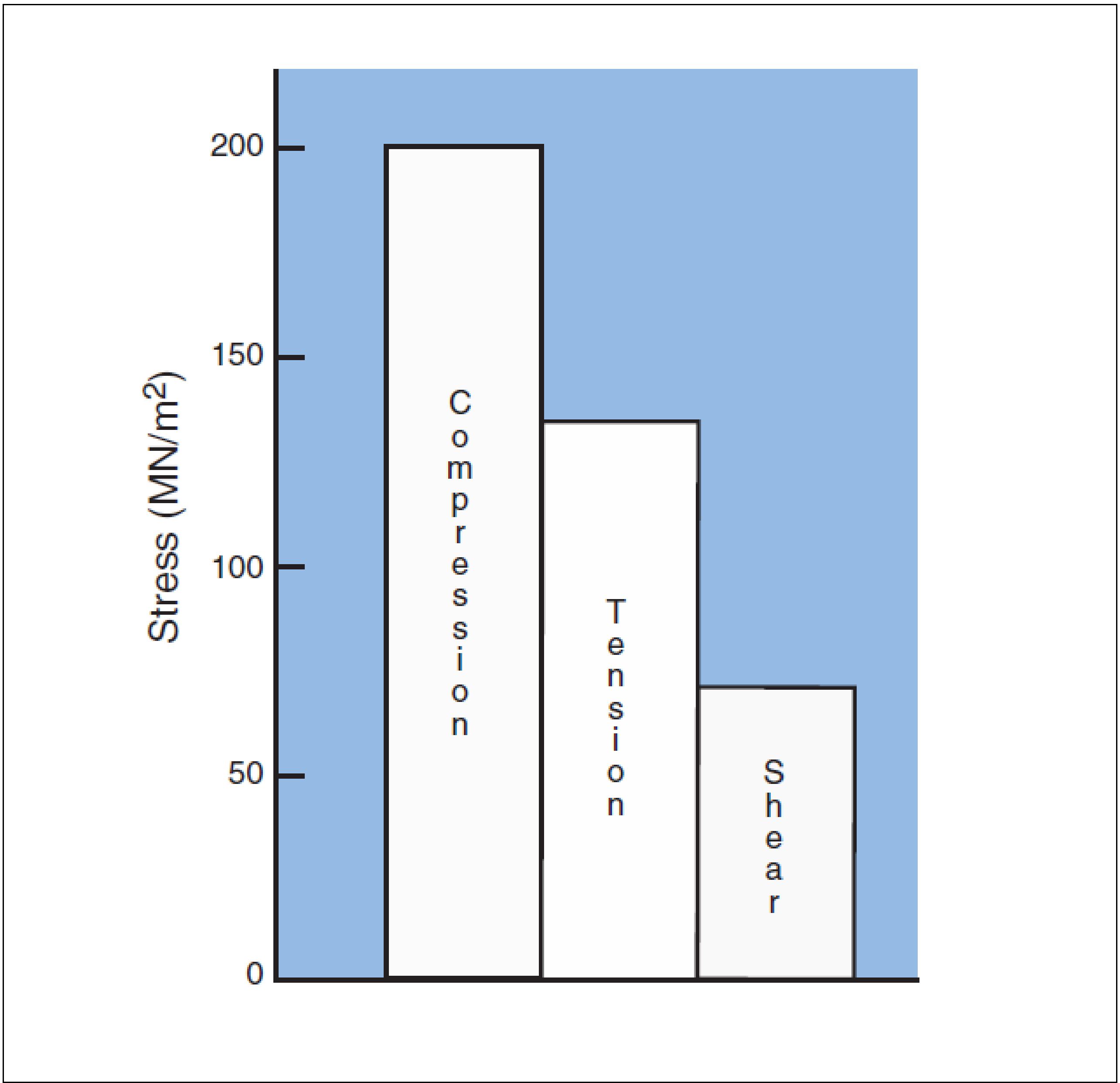


Figure 5-3: Ultimate stress for human adult cortical bone specimens

The compressive strength of cortical bone is greater than that of concrete, wood, or glass (Figure 5-4). The strength of cortical bone in the middle of long bones is demonstrated in the ability to tolerate large impact loads and resist bending. Cancellous bone strength is less than that of cortical bone, but cancellous bone can undergo more deformation before failure.

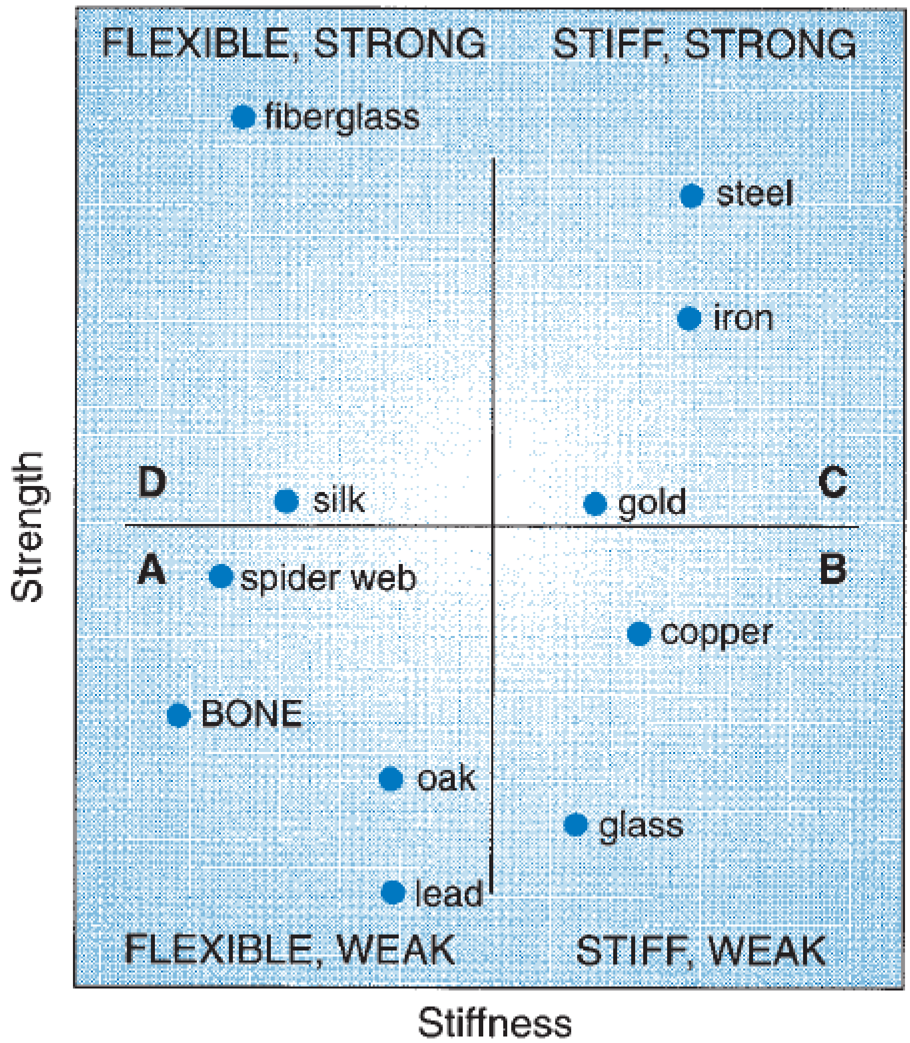


Figure 5-4: The strength and stiffness

Figure 5-4 show the strength and stiffness of a variety of materials are plotted in four quadrants representing material that is flexible and weak (A), stiff and weak (B), stiff and strong (C), and flexible and strong (D). Bone is categorized as being flexible and weak, along with other materials, such as spider web and oak wood.

5.3.1.2 Stiffness

Stiffness, or the modulus of elasticity, is determined by the slope of the load deformation curve in the elastic response range and is representative of the material's resistance to load as the structure deforms. The stress–strain curve for ductile, brittle, and bone material is shown in Figure 5-5, and Figure 5-4 plots a variety of materials according to strength and stiffness (49). Metal is a type of ductile material that has high stiffness, and at stresses beyond its yield point, exhibits ductile behavior in which it undergoes large plastic deformation before failure. Glass is a brittle material that is stiff but fails early, having no plastic region. Bone is not as stiff as glass or metal, and unlike these materials, it does not respond in a linear fashion because it yields and deforms nonuniformly during the loading phase (43). Bone has a much lower level of stiffness than metal or glass and fractures after very little plastic deformation.

At the onset of loading, bone exhibits a linearly elastic response. When a load is first applied, a bone deforms through a change in length or angular shape. Bone deforms no more than approximately 3% (50). This is considered in the elastic region of the stress–strain curve because when the load is removed, the bone recovers and returns to its original shape or length. With continued loading, the bone tissue reaches its yield point, after which its outer fibers begin to yield, with microtears and debonding of the material in the bone. This is termed the *plastic region* of the stress–strain curve. The bone tissue begins to deform permanently and eventually fractures if loading continues in the plastic region. Thus, when the load is removed, the bone tissue does not return to its original length but stays permanently elongated. Although bone can exhibit a plastic response, normal loading remains well within the elastic region. Bone behaves largely like a brittle material, exhibiting very little permanent plastic deformation to failure [18].

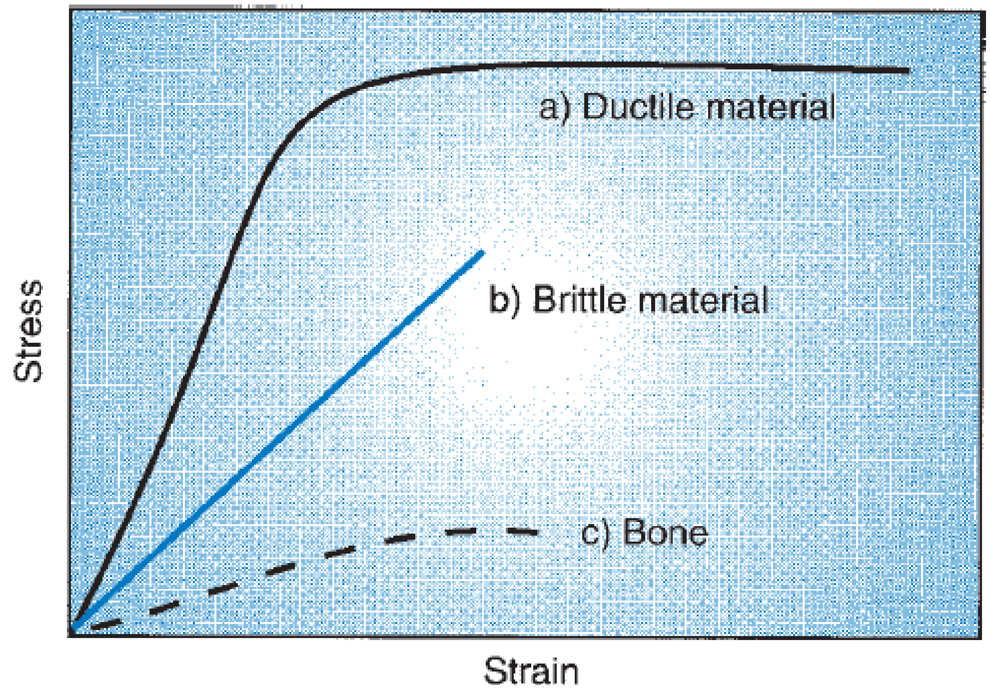


Figure 5-5: Stress–strain curves illustrating the differences in the behavior between ductile material (A), brittle material (B), and bone (C)

Stress–strain curves illustrating the differences in the behavior between ductile material (A), brittle material (B), and bone (C), which has both brittle and ductile properties. When a load is applied, a brittle material responds linearly and fails or fractures before undergoing any permanent deformation. The ductile material enters the plastic region and deforms considerably before failure or fracture. Bone deforms slightly before failure. Figure 5-5.

5.3.2 Viscoelastic Characteristics

Bone is viscoelastic, meaning that its response depends on the rate and duration of the load. At higher speeds of loading, bone becomes stiffer, tougher and it can absorb more energy before breaking. These strain rates are seen in high-impact situations involving falls or vehicular accidents. As shown in Figure 5-6, a bone loaded slowly fractures at a load that is approximately half of the load handled by the bone at a fast rate of loading.

Bone tissue is a viscoelastic material whose mechanical properties are affected by its deformation rate. The ductile properties of the bone are provided by its collagenous material. The collagen content gives the bone the ability to withstand tensile loads. Bone is also brittle, and its strength depends on the loading mechanism. The brittleness of bone is provided by the mineral constituents that provide bone with the ability to withstand compressive loads [18].

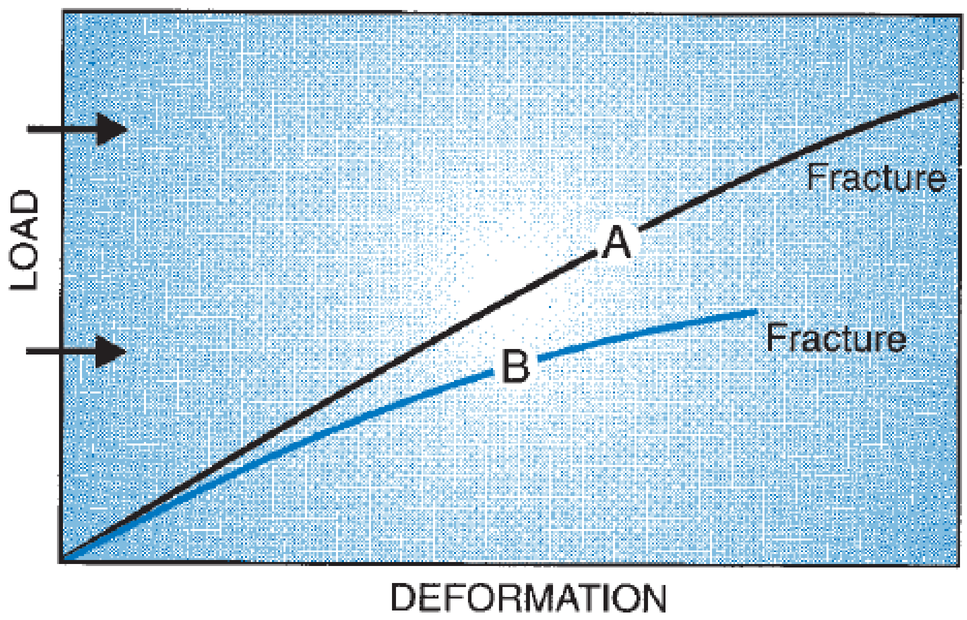


Figure 5-6: Bone is considered viscoelastic

Bone is considered viscoelastic because it responds differently when loaded at different rates. (A) When loaded quickly, bone responds with more stiffness and can handle a greater load before fracturing. (B) When loaded slowly, bone is not as stiff or strong, fracturing under lower loads. Figure 5-6.

5.4 Loads Applied to Bone

The skeletal system is subject to a variety of applied forces as bone is loaded in various directions. Loads are produced by weight bearing, gravity, muscular forces, and external forces. Internally, loads can be applied to bones through the joints by means of ligaments or at tendinous insertions, and these loads are usually below any fracture level. Externally, bone accommodates multiple forces from the environment that have no limit on magnitude or direction.

Muscular activity can also influence the loads that bone can manage. Muscles alter the forces applied to the bone by creating compressive and tensile forces. These muscular forces may reduce tensile forces or redistribute the forces on the bone. Because most bones can handle greater compressive forces, the total amount of load can increase with the muscular contribution. If muscles fatigue during an exercise bout, their ability to alleviate the load on the bone diminishes. The altered stress distribution or increase in tensile forces leaves the athlete or performer susceptible to injury [18][19].

The stress and strain produced by forces applied to bones are responsible for facilitating the deposit of osseous material. Stress can be perpendicular to the plane of a cross section of the loaded object. This is termed **normal stress**. If stress is parallel to the plane of the cross section, it is termed **shear stress**. Each type of stress produces a strain. For example, whereas normal strain involves a change in the length of an object, shear strain is characterized by a change in the original angle of the object. An example of both normal strain and shear strain is the response of the femur to weight bearing. The femur shortens in response to normal strain and bends anteriorly in response to shear strain imposed by the body weight (46). Normal stress and shear stress, developed in response to tension applied to the tibia, are presented in Figure 5-7. Normal and shear strain, developed in response to compression of the femur, are also illustrated [18].

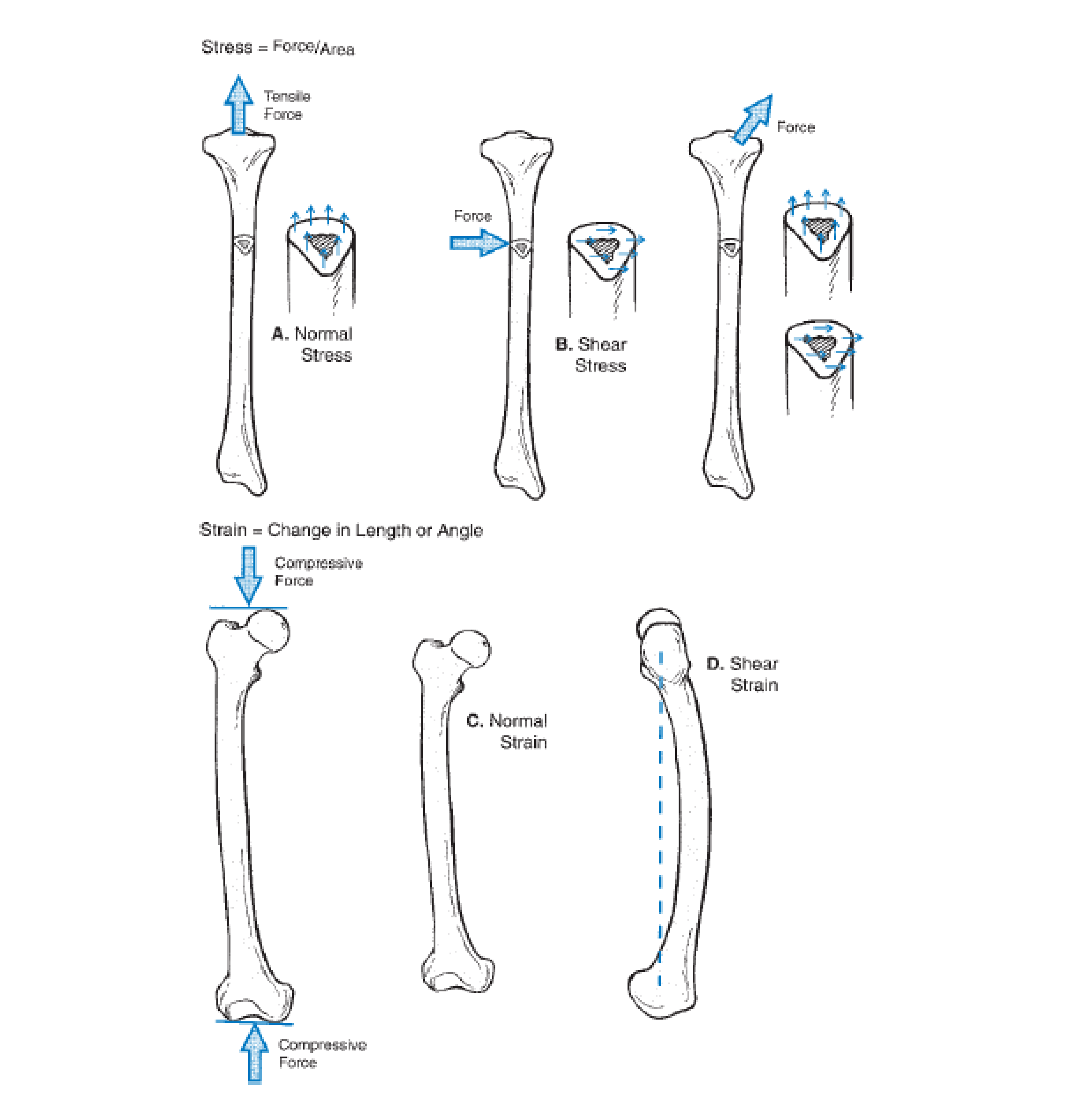


Figure 5-7: Stress, or force per unit area

Stress, or force per unit area, can be perpendicular to the plane (normal stress) (A) or parallel to the plane (shear stress) (B). Strain, or deformation of the material, is normal (C), in which the length varies, or shear (D), in which the angle changes. Figure 5-7.

Whether or not a bone incurs an injury as a result of an applied force is determined by the critical strength limits of the material and the loading history of the bone. External factors related to fracture include the magnitude, direction, and duration of the force coupled with the rate at which the bone is loaded. The ability of a bone to resist fracture is related to its energy-absorbing capacity. The ability of a bone to resist deformation varies through its length because of the different makeup of cortical and cancellous bone (38). Cancellous bone, depending on its architecture, can deform more and can absorb considerably more energy than cortical bone (38). These limits are primarily influenced by the loading on the bone. The loading of the bone can be increased or decreased by physical activity and conditioning, immobilization, and skeletal maturity of the individual. The rate of loading is also important because the response and tolerance of bone is rate sensitive. At high rates of loading, when bone tissue cannot deform fast enough, an injury can occur [18].

Table 5-1: Different Types of Loads Acting on Bone

Load	Type of Force	Source	Stress/Strain
Compression	Presses ends of bones together to cause widening and shortening	Muscles, weight bearing, gravity or external forces	Maximal stress on the plane perpendicular to the applied load
Tension	Pulls or stretches the bone to cause lengthening and narrowing	Usually pull of contracting muscle tendon	Maximal stress on the plane perpendicular to the applied load
Shear	Force applied parallel to surface, causing internal deformation in an angular direction	Compressive or tension force application or external force	Maximum stress on the plane parallel to the applied load
Bending	Force applied to the bone having no direct support from the structure	Weight bearing or multiple forces applied at different points on the bone	Maximum tensile forces on the convex surface of the bent member and maximum compression forces on the concave side
Torsion	Twisting force	Force applied with one end of the bone fixed	Maximum shear stress on both the perpendicular and parallel to axes of bone with tension and compression forces also present at an angle across the surface

The five types of forces applying loads to bone are compression, tension, shear, bending, and torsion. These forces are summarized in Table 5-1 and illustrated in Figure 5-8.

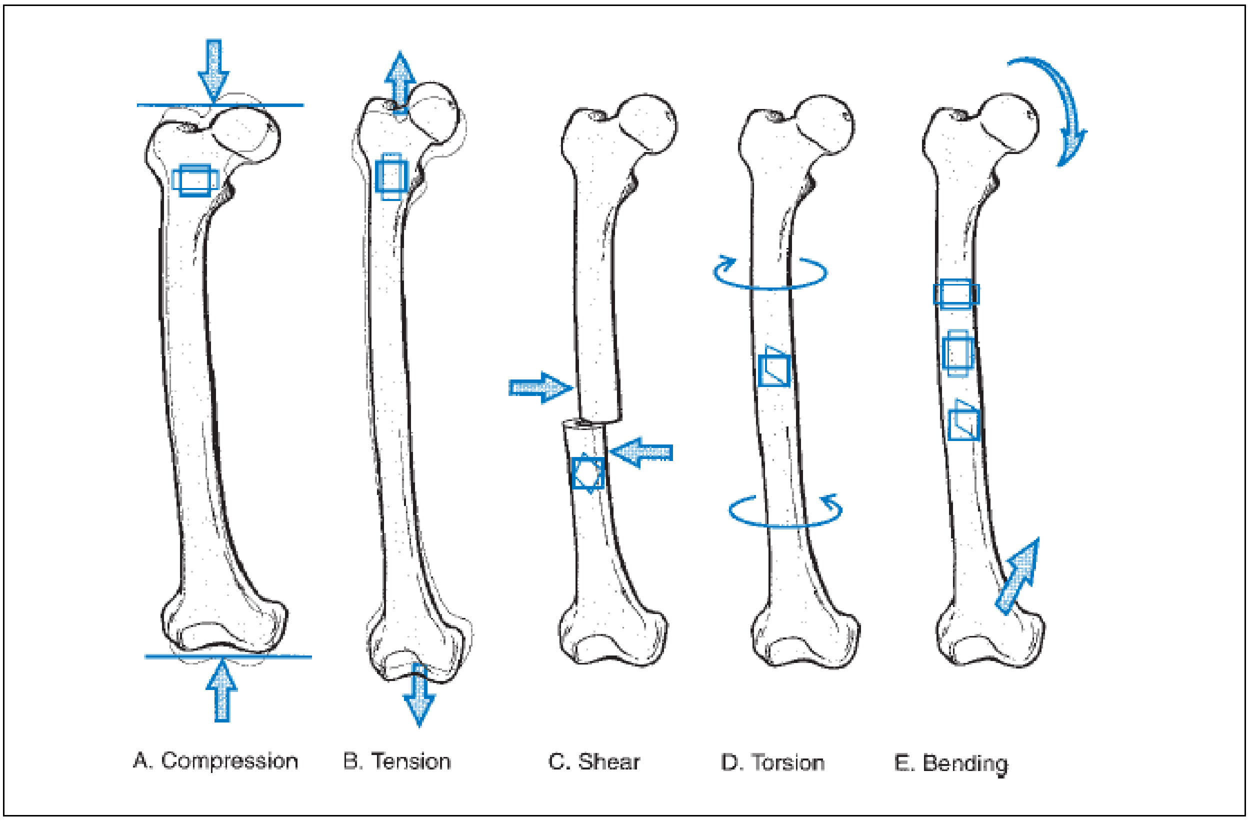


Figure 5-8: The skeletal system is subjected to a variety of loads that alter the stresses in the bone

The skeletal system is subjected to a variety of loads that alter the stresses in the bone. The *square* in the femur indicates the original state of the bone tissue. The *colored area* illustrates the effect of the force applied to the bone. (A) Compressive force causes shortening and widening. (B) Tensile force causes narrowing and lengthening. (C) and (D) Shear force and torsion create angular distortion. (E) Bending force includes all of the changes seen in compression, tension, and shear. Figure 5-8.

CHAPTER SIX

Methodology And Results

6.1 Introduction

6.2 Processes of Geomagic Design X for an object

6.3 Export the development object to CATIA

6.4 Model Analysis using ANSYS

6.5 Results

6.6 Conclusion

6.7 Recommendation and Future work

6.1 Introduction

In the first we should confirm that we could not get on the femur Because of the lack of availability and the difficulty of procedures to obtain them So we have replaced our work with Cows bone.

However, there are a number of problems that we encountered during the period of bone acquisition because fat and organic matter adhere to bone and was solved as follows:

We worked on boiling the bone into the water to get rid of fat and dissolve It, but this method failed due to the expansion of the material heat and damage the piece and then we work in the second method is the process of removing fat using vinegar, because of the acidic materials and after removal of fat and organic matter we obtained the following form shown in Figure 6-1.

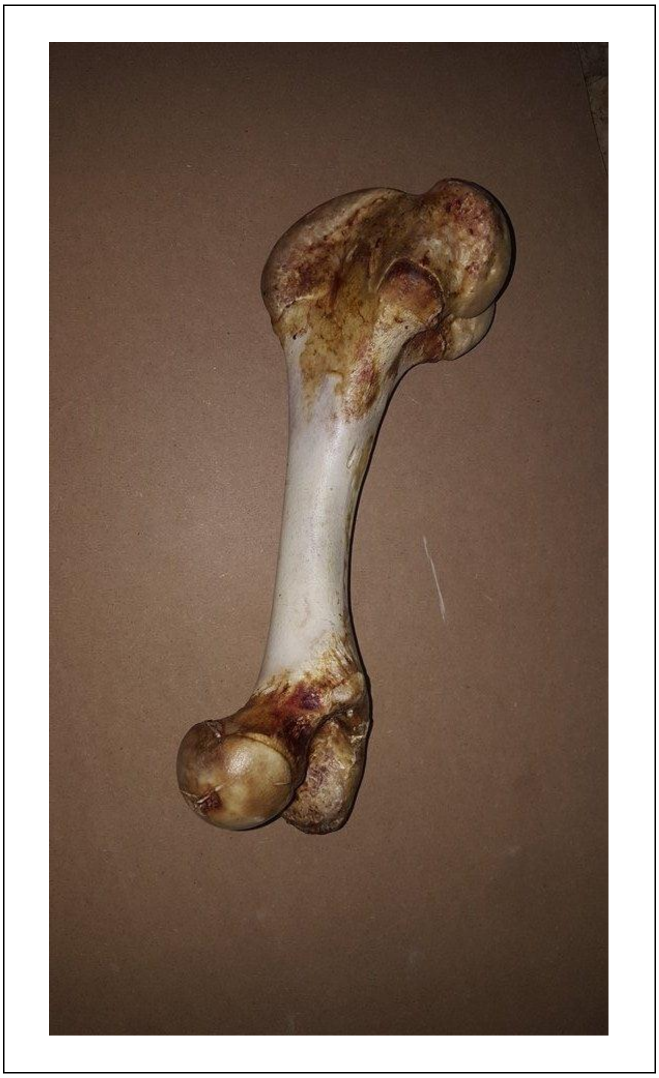


Figure 6-1: Original form of cow femur

3D Scanning Data shown Figure 6-2.

Figure 6-2: 3D Scanning cow femur bone

6.2 Processes of Geomagic Design X for an object

This processes is starting after obtaining a scanning data of sample, first step starts by import the scanning file data as in Figure 6-3, usually the type of this files is “.xrl “, also these files are in mesh data, Figure 6-4 shows the basic sample data that this project works on it.

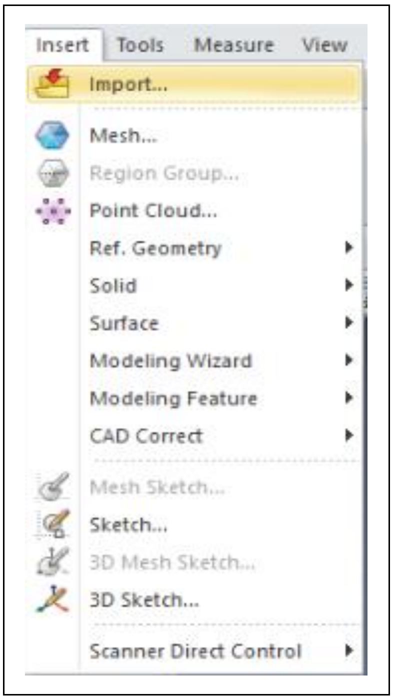


Figure 6-3: Import ".xrl" File

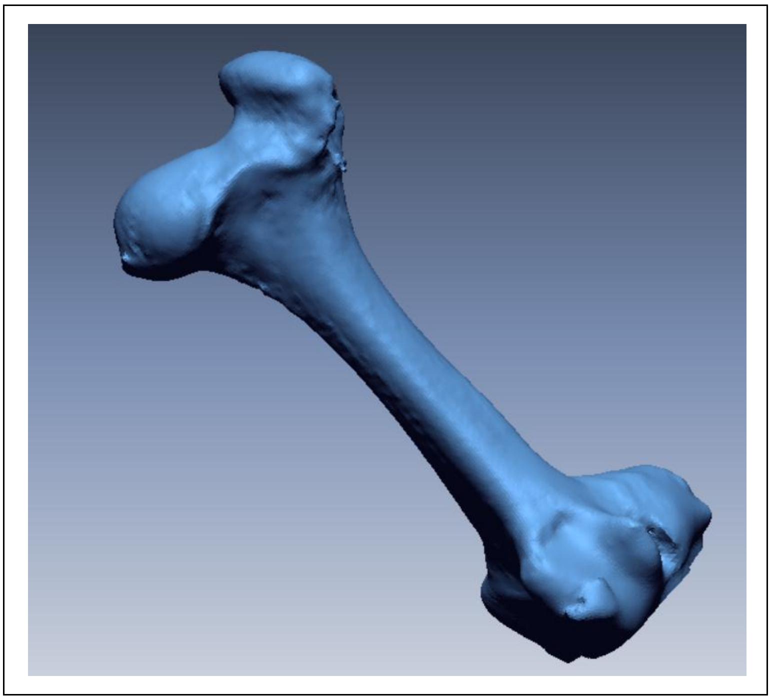


Figure 6-4: Original form of cow femur

6.2.1 Mesh Build up Wizard

After import the scanning data, this is the first step for optimizing this phase, through this command can wizard style inter face for creating defect-free, also it help when the 3D scan data files that have not been aligned This tool consists of 5 stages. Each stage can be executed in just few clicks, enabling the speedy creation of optimal mesh for use in the final reverse design stages. The 5 stages are [1]:

- Data Preparation stage: Analyses the state of scan data and defining the next process according to the type of target scan data.
- Data Editing stage: Remove noisy clusters and unnecessary point clouds or meshes
- Data Pre-Aligning stage: Aligns multiple scan data files quickly.
- Best-Fit Aligning stage: Aligns multiple scan data files by using geometry shape information.
- Data Merging stage: Merges scan data and creates an optimized mesh.

6.2.2 Optimize the Meshing data by Mesh mode

This process will start by select the object from the feature tree, then applying Mesh mode, table 6-1 shows an important tools icon that used in this mode to obtain a specific surface.

Table 6-1: Tools in mesh phase

Symbol	Tool Name
	Healing Wizard
	Optimize Mesh
	Smooth
	Global Remesh
	Defeature

6.2.2.1 Healing wizard

By using **healing wizard**, the object automatically detects various defects in the mesh such as Non-manifold poly vertices, fold, small, crossing, non-manifold and dangling poly faces, small clusters and small tunnels. So, can be obtained mesh model without any abnormal poly-faces, Figure 6-5 shows a femur cow before healing wizard

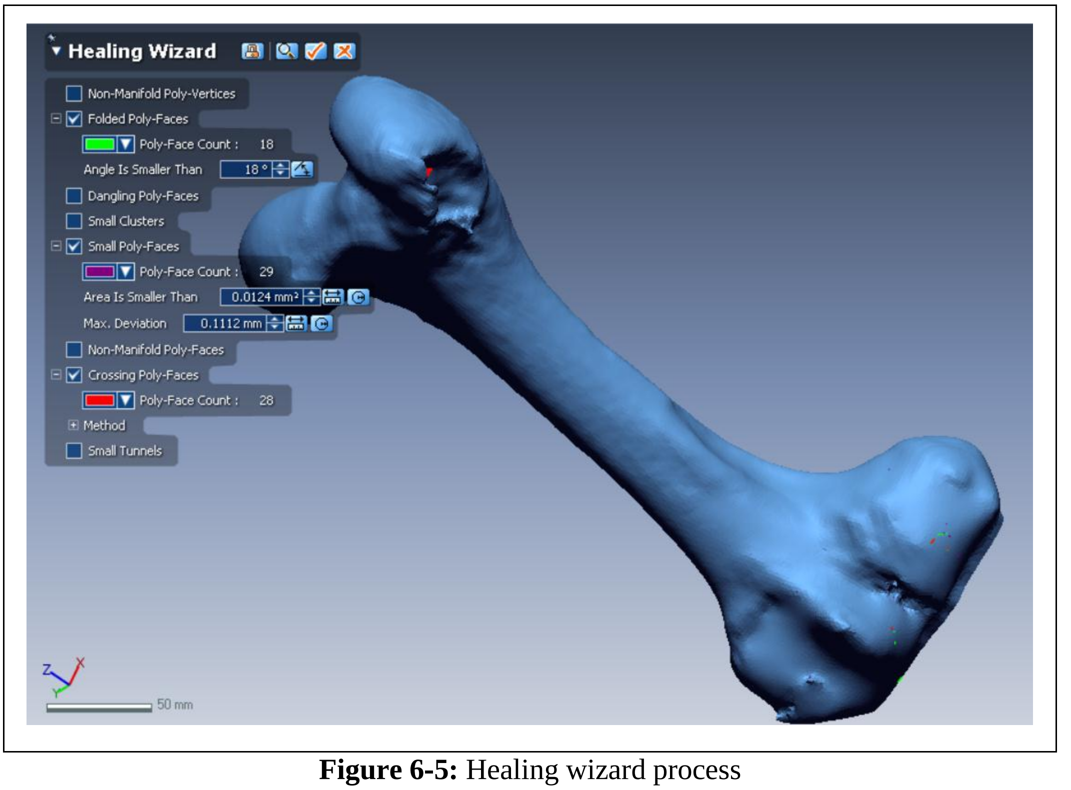


Figure 6-5: Healing wizard process

6.2.2.2 Smooth

After the healing wizard processes, there is a lot of optimizing on 3D scanning data but the mesh still has some defects, as in Figure 6-6 left side, mesh seems so rough and contains some of noises, in this shape there is not clear, but when making a scan for very shiny object, the reflection generates noise, and generate a rough surface on the mesh. **Smooth** tool removes spiky points by averaging with surrounding points and improving the quality of a mesh.

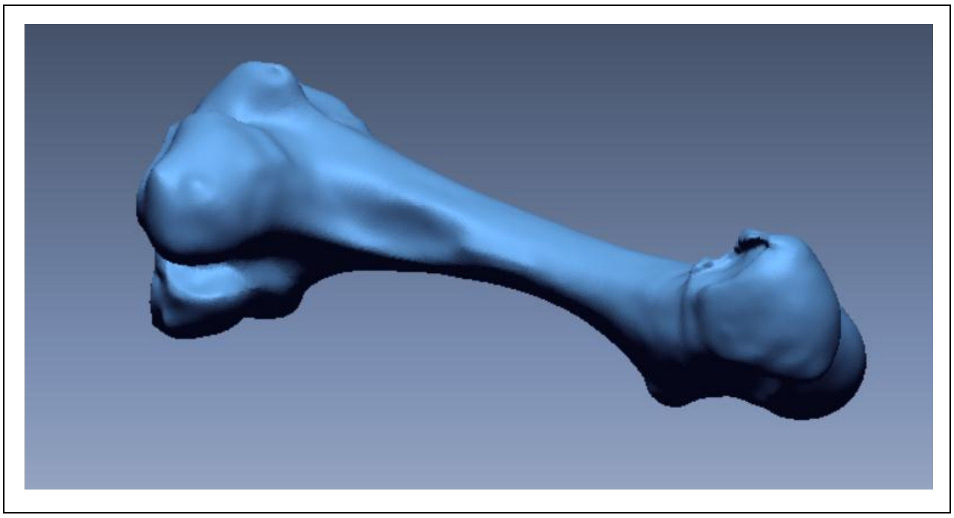


Figure 6-6: After smooth process

6.2.2.3 Smart Prush

The smart brush command partially optimizes a target mesh and improves the quality of the mesh. This command uses advanced methods to interactively edit partial areas of mesh such as smooth, decimate, clean, enhance and deform.

The smart brush command is useful for:

1. Partially optimizing and improving the quality of mesh.
2. Reducing the number of poly-faces in a specify area of a mesh.
3. Partially deforming mesh.

6.2.2.4 Rewarp

If the scan data has many holes and wish to quickly auto surface the model the Rewrap tool will speed this process. Shown in Figure 6-7 and Figure 6-8.

The Rewrap command is useful for:

1. If the mesh has curved areas remove them using Region Groups. Use the Select by Region Group. These holes can be added later in the design process.
2. Create cross sections using the Section Tool in the 3D Mesh Sketch group. Or construct a patch network for a Boundary fit command.
3. Rebuild the Mesh Sketches with fewer intermediate points, and add a guide curve between the sketches.
4. Preform a Surface Loft between the sketches.
5. With high areas of curvature the loft may not represent the mesh accurately. To compensate for this refit the surface to the mesh with the Refit command.

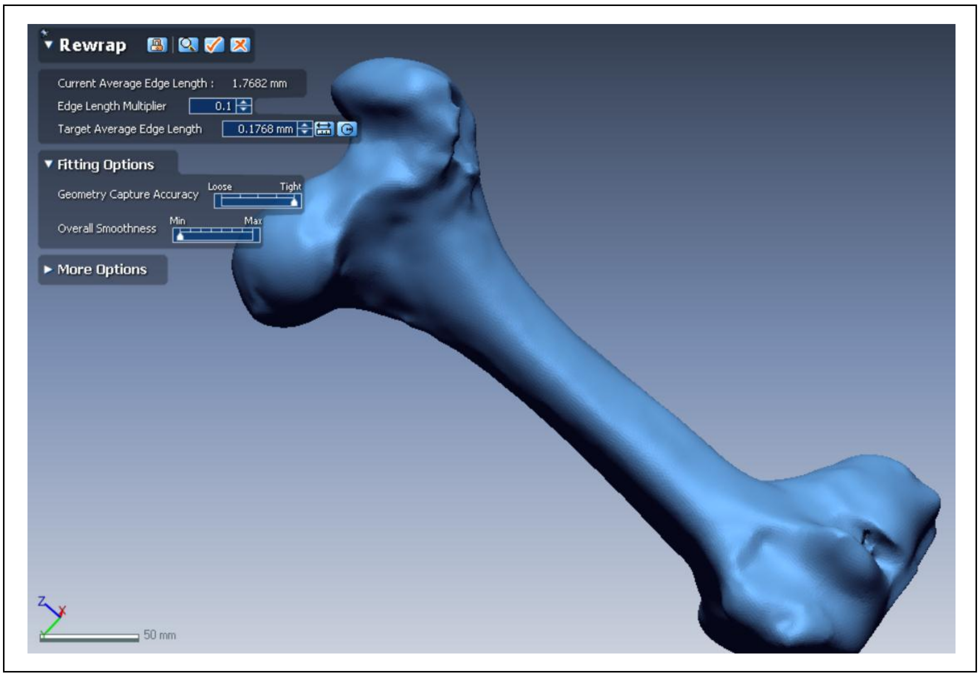


Figure 6-7: Before Rewarp

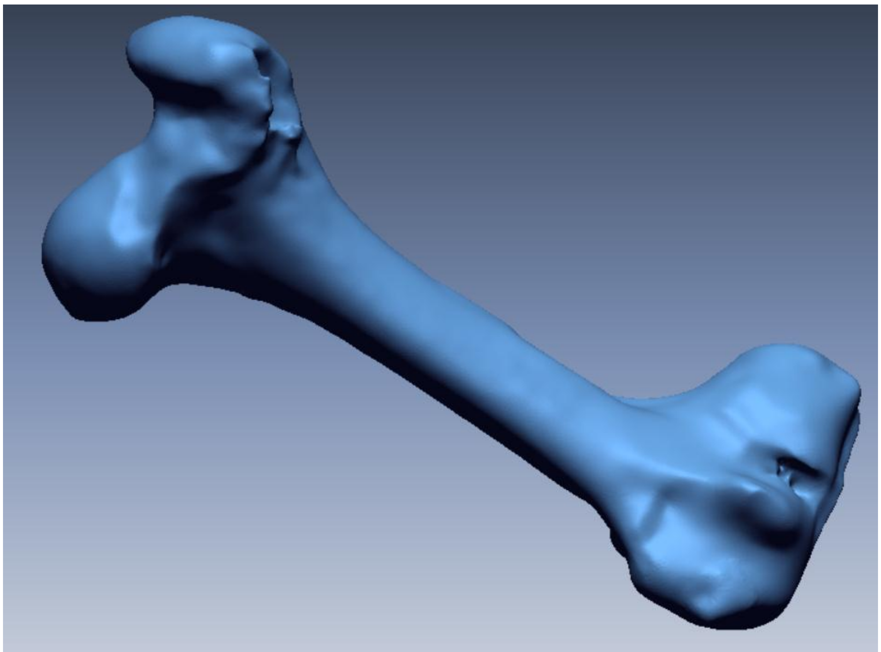


Figure 6-8: After Rewarb

6.2.2.5 Global remesh

The global remesh command globally regenerates poly-faces with uniform poly-edge lengths on a mesh and improves the quality of the mesh. this command also cleans and manifolds solid mesh that has no boundaries.

The Global remesh command is useful for:

1. Optimizing mesh so that it has poly-faces globally with uniform poly-edge lengths.
2. Optimizing mesh to be used for creating a high quality surface body.
3. Reducing the effects of defects and roughness on a mesh.
4. Creating a high quality mesh from combined mesh that is complex and overlapped.

6.2.2.6 Auto Surfacing

Auto surfacing technology that automatically converts point clouds into NURBS surface models has been developed by previous tools, during this command, region group phase will convert to surface body smooth and quickly, this tool uses when the shape is complex freeform part, it can create a surface body that can envelope an entire geometric shape of a specific mesh, the Figure6-9 shows the form of sole after applying auto surface.

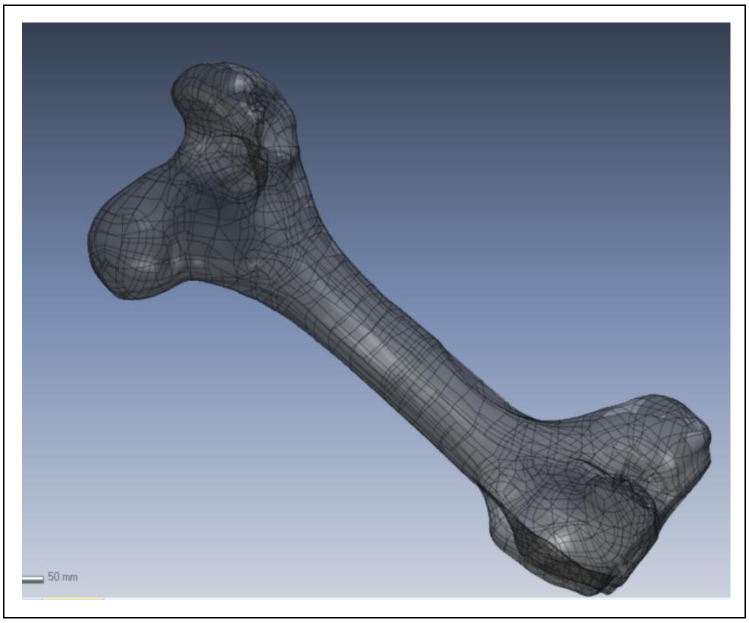


Figure 6-9: Auto surfacing

6.3 Export the development object to CATIA

Through this step can export the object to the CATIA or SolidWorks into Many multi-platform formats are offered, in order to enhancing if required or to create the core and cavity for the object, this command can be done by selecting the command Export from File tape, select the object, press OK, and save it in a specific folder as shown in Figure 6-10 and select an acceptable type format, that can other software deal with it, for CATIA V5 R20 for example, the suitable format is STEP file " .stp ".

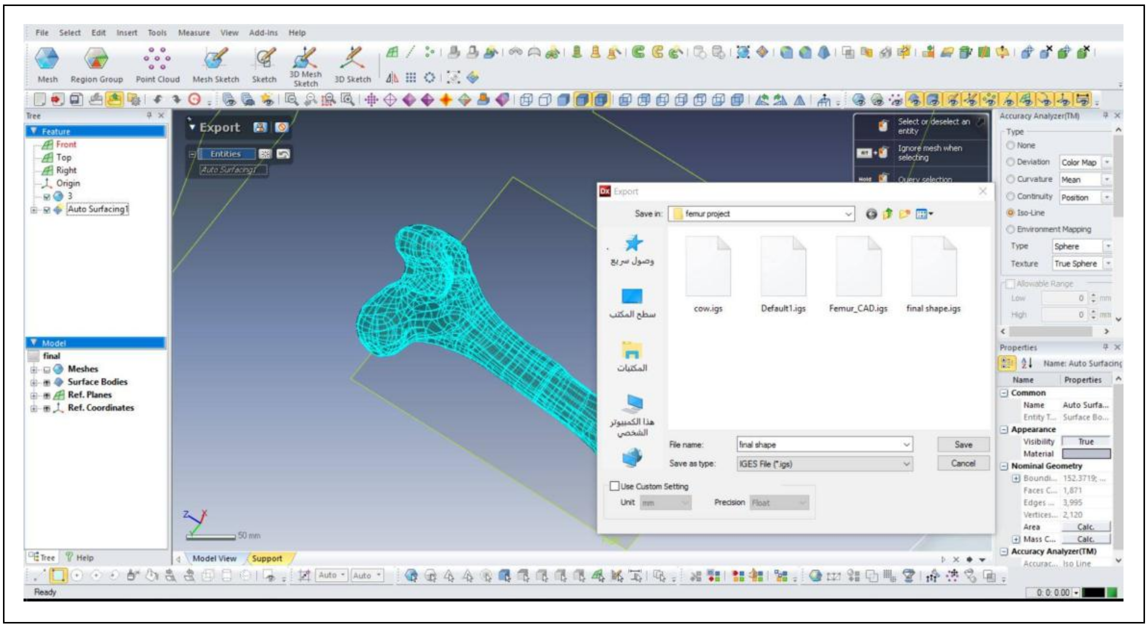


Figure 6-10: Export CATIA File ".igs"

6.4 Model Analysis using ANSYS

In this section will be completed the work on the human femur bone using ANSYS as obtained from an electronic source, shown in Figure 11.

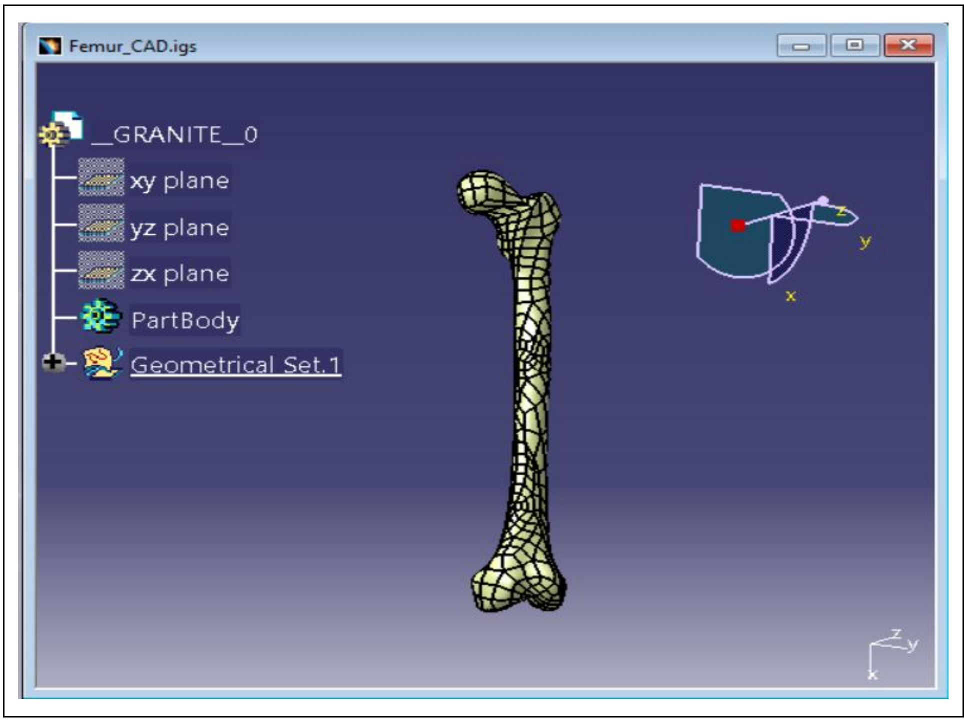


Figure 6-11: Human Femur Bone (CATIA)

6.4.1 Femur bone processing by ANSYS

The femur bone model will be analyzed as a static structure in order to analyze the model in standing position, shown in figure 12.

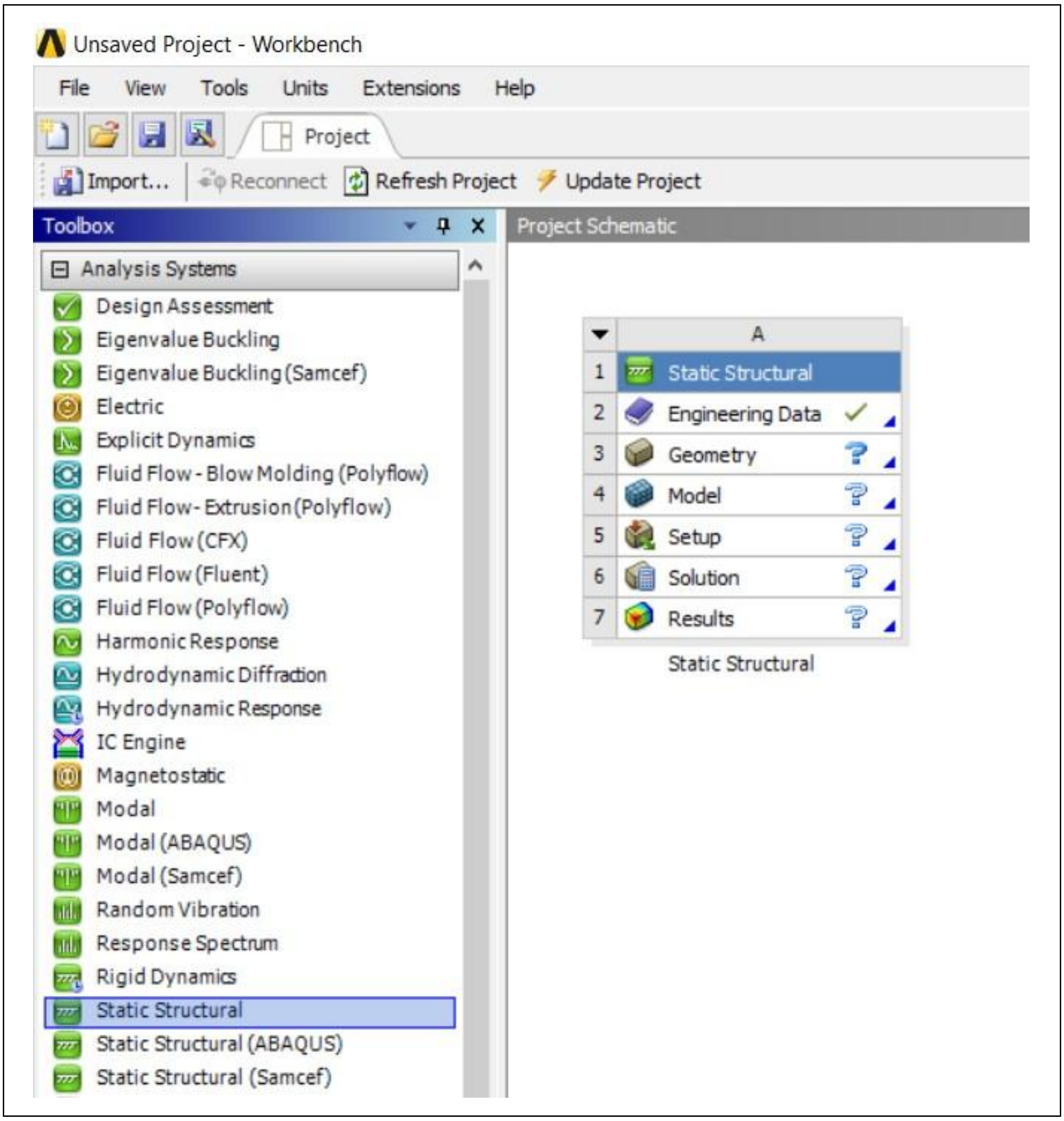


Figure 6-12: Static Structure

6.4.2 Material Assignments (Engineering Data)

Human bone is highly heterogeneous and nonlinear in nature, so it is difficult to assign material properties along each direction of bone model. In biomechanics study, material can be assign in two ways, either in Mimics (Materialise Interactive Medical Image Control System) or in Finite element module. Here material properties are directly assigned in ANSYS by using finite element module. The table (6_1), value 1 [4], value 2 [20], show properties of material; Density, Young's Modulus, Poisson's Ratio and shear modulus.

Table 6-2: Properties of material

Parameter		**Value 1**	**Value 2**
Density(g/cm^3)		2.05	2.0208
young's modulus (MPa)	x	1940	6982.9
	y	1250	6982.9
	z	1250	18155
Poisson's Ratio	v_{xy}	0.33	0.4
	v_{yz}	0	0.25
	v_{xz}	0	0.25
Shear Modulus (MPa)	xy	5700	4690
	yz	4850	5610
	xz	5700	7680

6.4.3 Import Geometry

This processes is starting after obtaining a CATIA data, first step starts by import the CATIA file data as in Figure 6-13, usually the type of this files is ".igs ", and edit model shown in Figure 6-14.

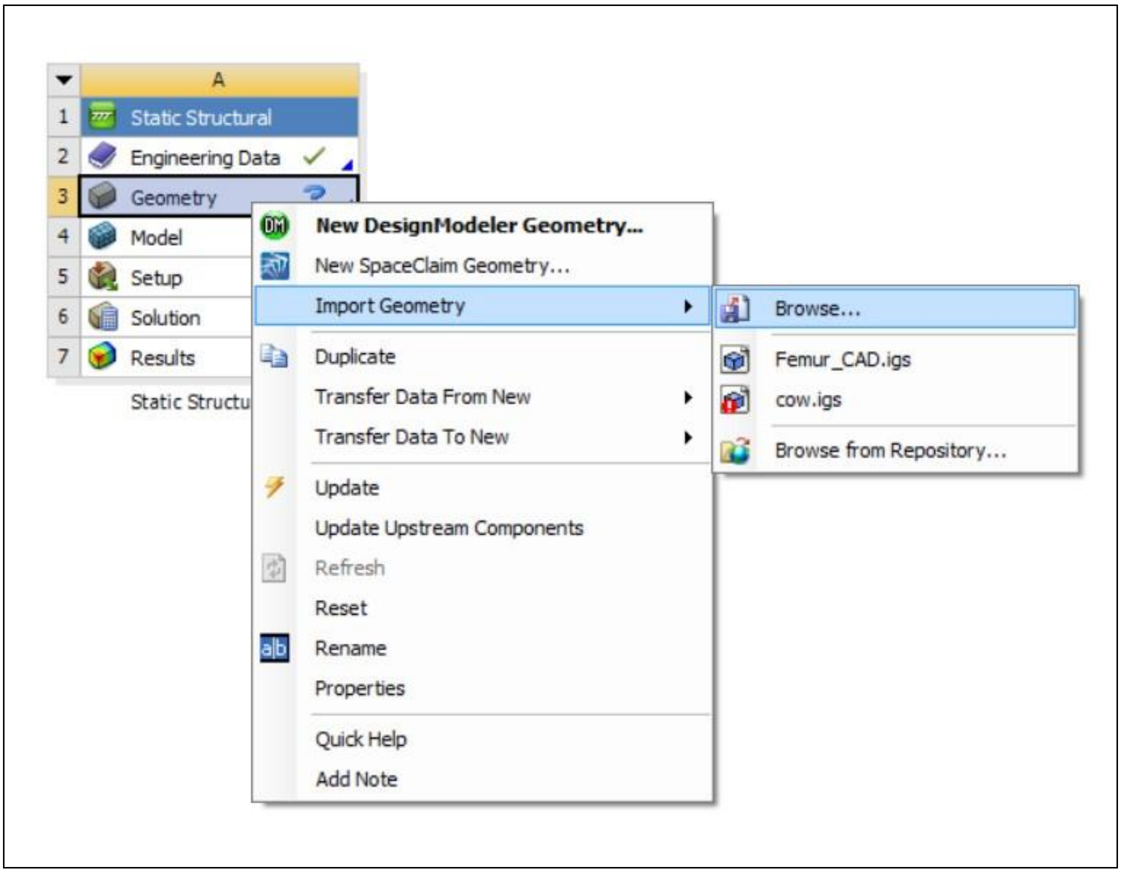


Figure 6-13: Import the CATIA file data

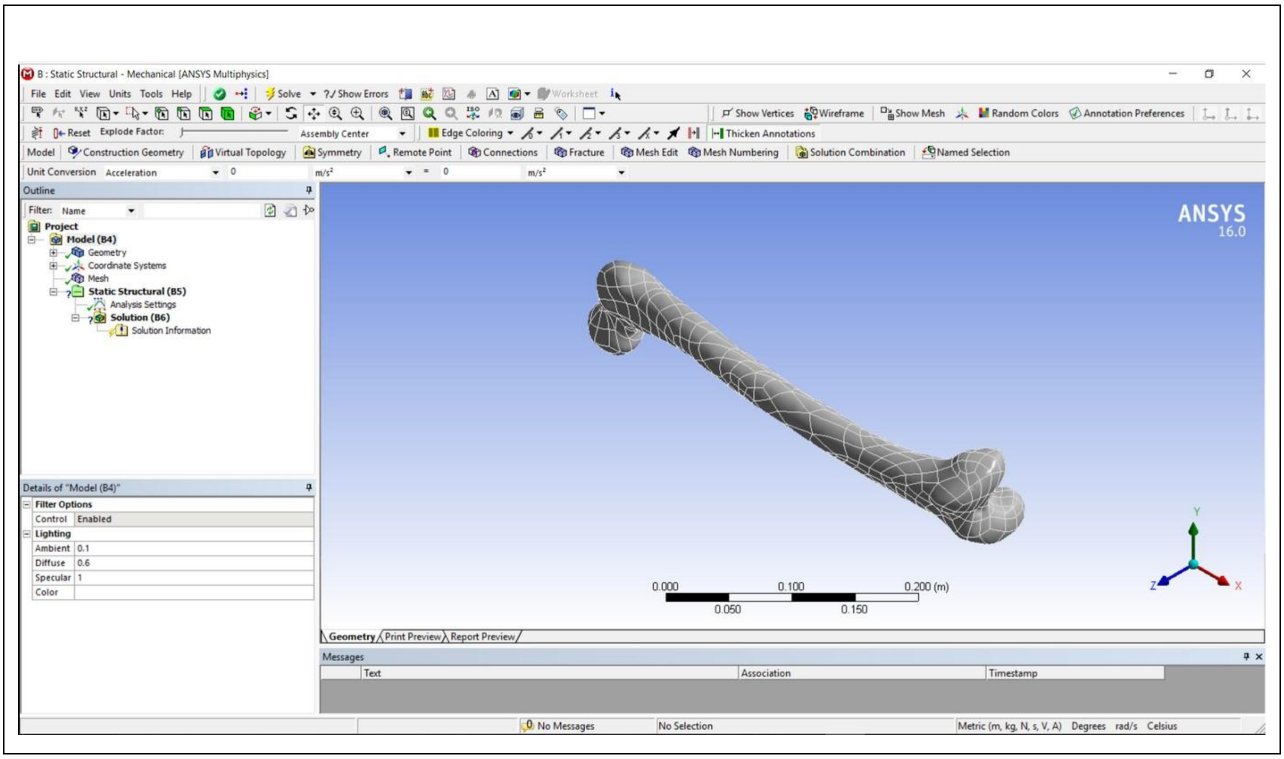


Figure 6-14: Femur Model

6.4.4 Generation Mesh

Mesh is a very important step required for Finite Element Analysis of the femur model, an optimized mesh has been developed using model wizard in ANSYS Workbench. In automatic remeshed operation surface mesh of equilateral triangle is generated a proper setting and values have been executed in order to use smaller elements on proximities and curvatures for the model. The numbers of tetrahedral elements used for the femur model are 2400, while the number of nodes is 4581 as shown in Figure 6-15.

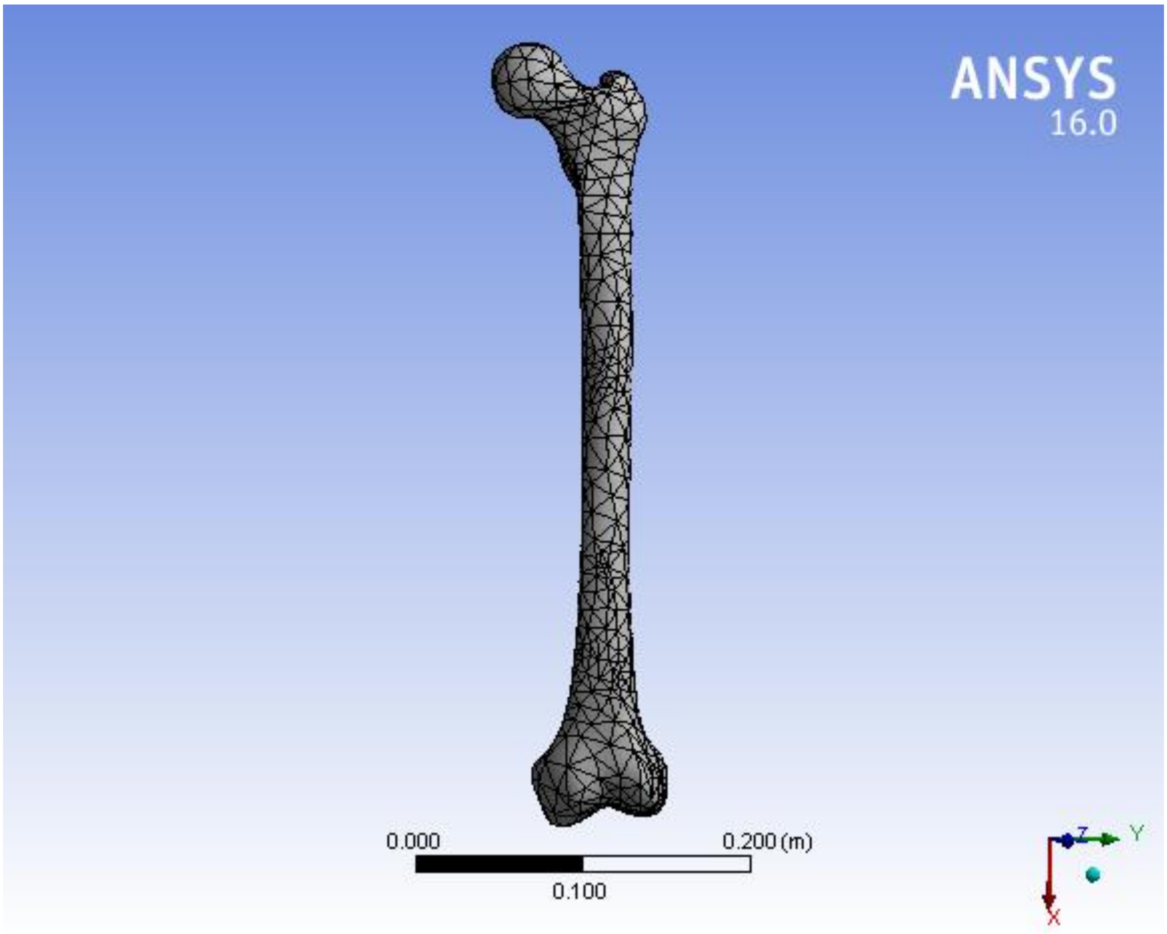


Figure 6-15: The femur model after making mesh process

6.4.5 Boundary conditions

Femur bone is solid and inflexible. The three dimensional Finite element model of femur bone with Surface mesh was imported in ANSYS. Since the femur bone model is nonlinear and highly heterogeneous in nature, model is first imported in Finite Element Modeler then transfer to static structural module in ANSYS 16.0. An eccentric and concentrate load of 750 Pa applied at the head of femur bone and fixed support is provided at lateral condyle, medial condyle and patellar surface. The boundary conditions are shown in Figure 6-16.

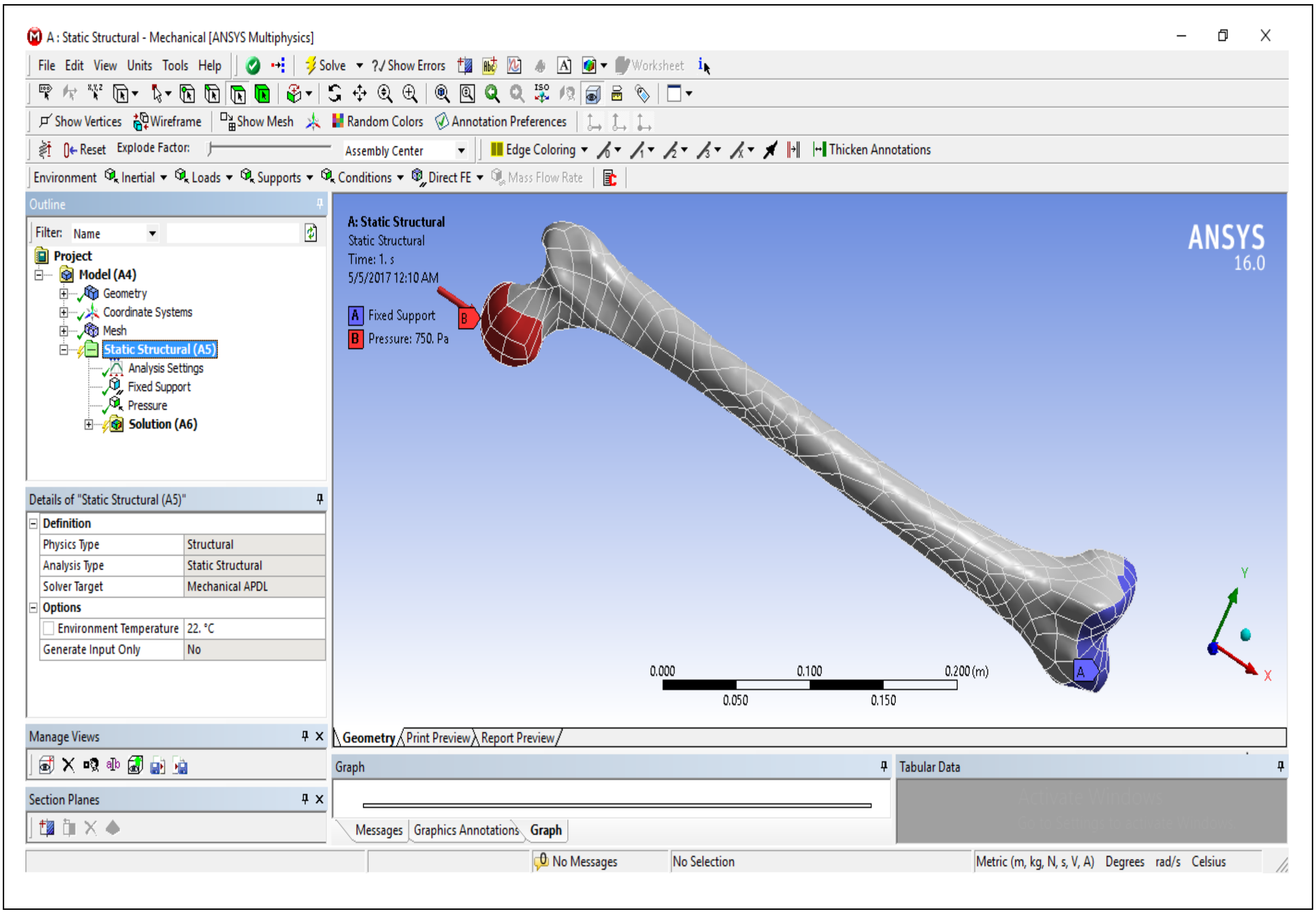


Figure 6-16: Boundary Conditions applied on Femur bone

6.5 Results

The Three dimensional FE model of femur bone generated from scanning data have become interest because of their high Potential in clinical practice. Here an automatic mesh generator provides very fine, good and fast geometrical representation of femur with accuracy. The entire process requires a couple of hours. It may be noted that only static load applied on femur. The results depend on the accuracy of FE model with reference to real conditions. This study investigates stress distribution, total deformation and fatigue failure of femur for a weight of 75 Kg during normal position. For eccentric load maximum total deformation 0.020143 m was obtained Figure 6-17. Results shows that higher deformation occurs at the head of femur and lowest occur at the lower end. Maximum principle stress 31760 Pa and Minimum principle stress is -722.7 Pa Figure 6-18. Maximum principle stress is generated at the middle section of the femur. The max equivalent (Von Mises) stress 31717 Pa occurs. Figure 6-19. Fatigue life was obtained 1×10^9 Figure 6-20, and factor of safety is constants throughout the femur were 15 Figure 6-21. This model investigates that the mechanical properties vary across the femur bone under physiological conditions and also with individuals. The Normal stresses obtained from the stress analysis of the human femur bone during standing up are shown in Figure 6-22, the maximum normal stress during normal standing up is 8.9221 MPa maximum normal stress during standing up is 6.9 MPa. The results show that higher weight provides higher total displacement. This model is useful for surgeon in femur surgeries and bone prosthesis.

Appendix A show the details and curve of experiment.

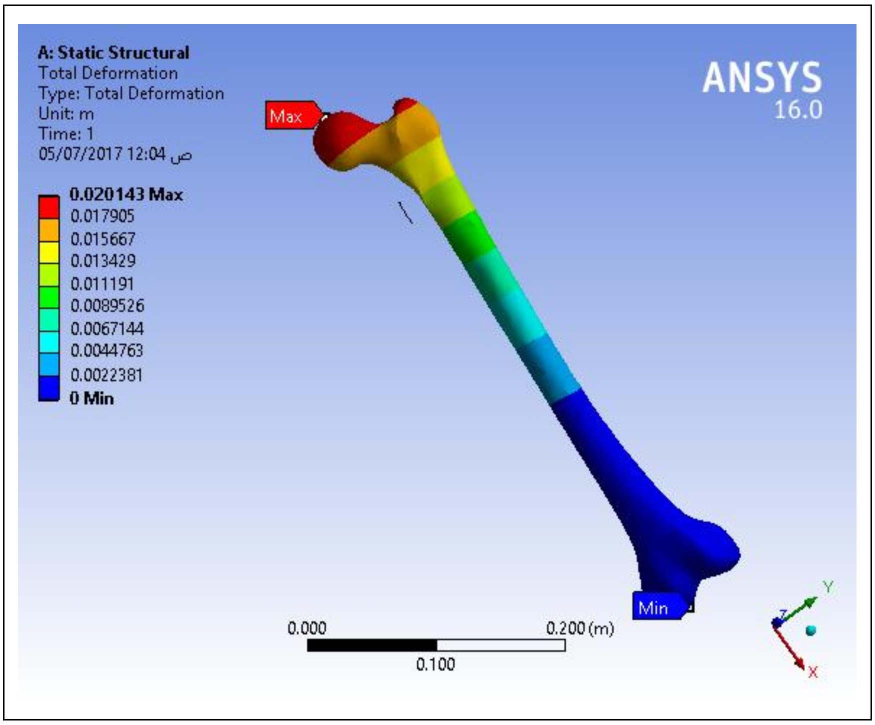


Figure 6-17: Total deformation of femur bone

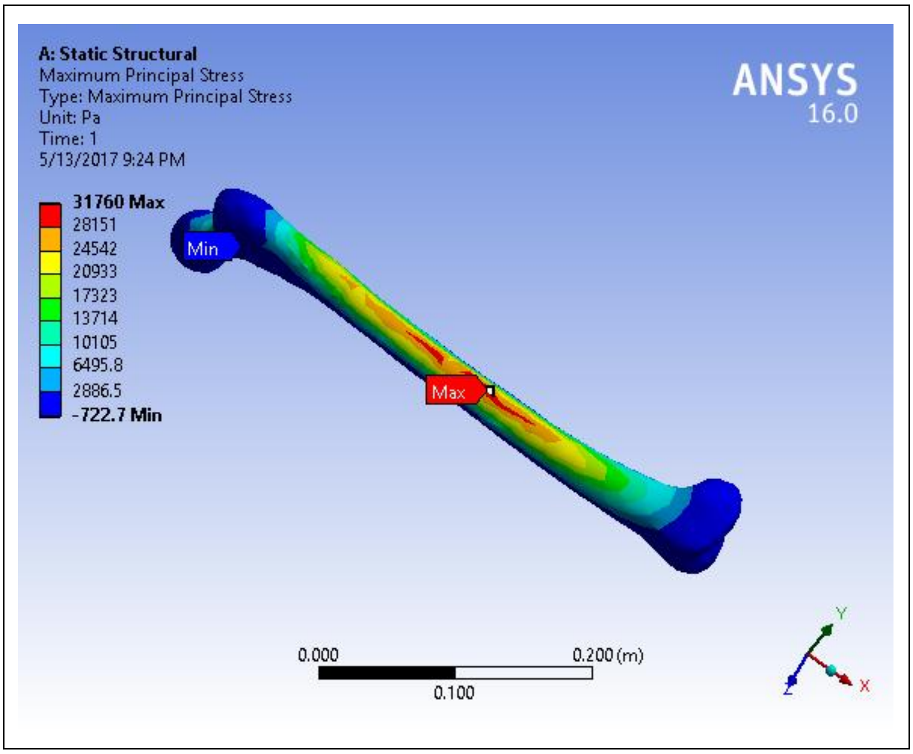


Figure 6-18: Principal stress in femur bone

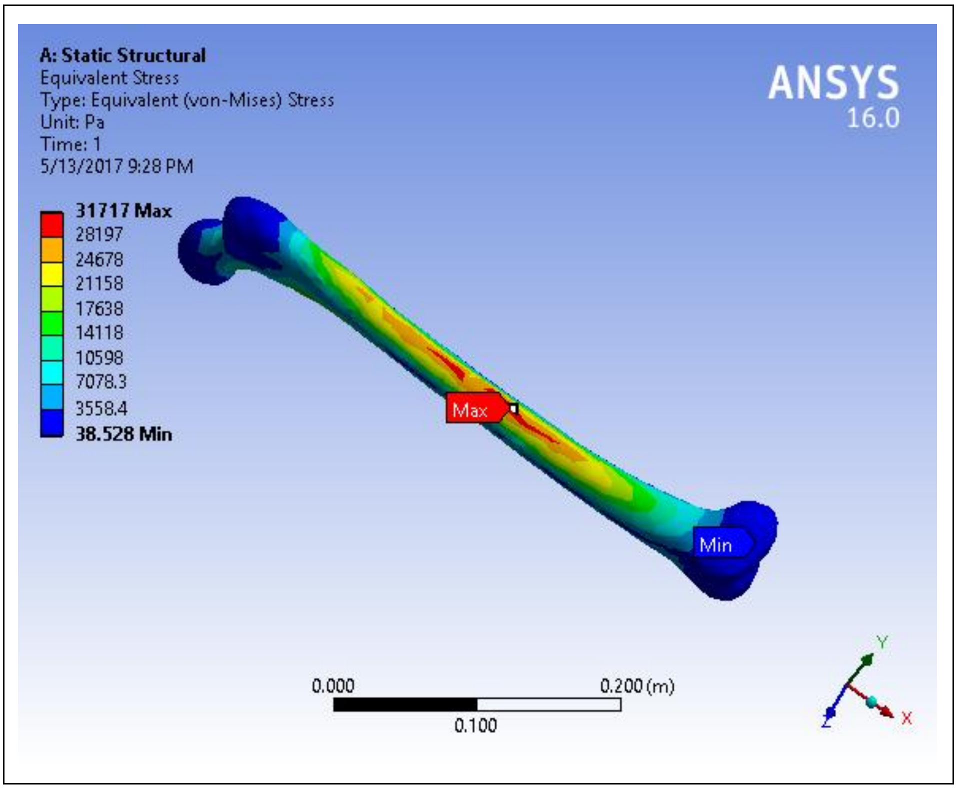


Figure 6-19: Equivalent (Von Mises) Stress

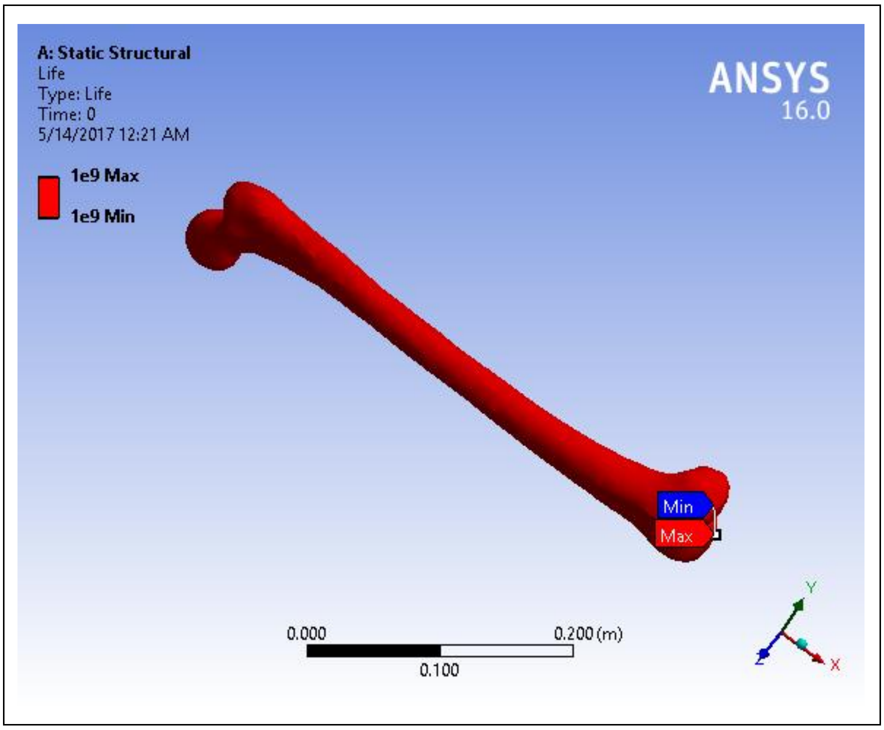


Figure 6-20: Fatigue Life

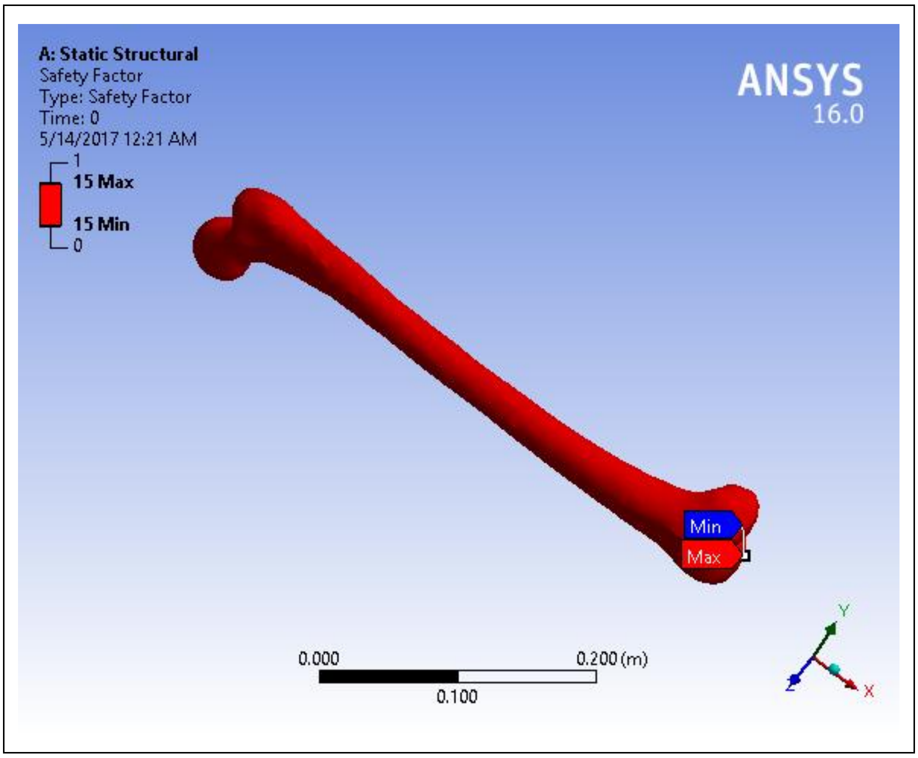


Figure 6-21: Safety factor

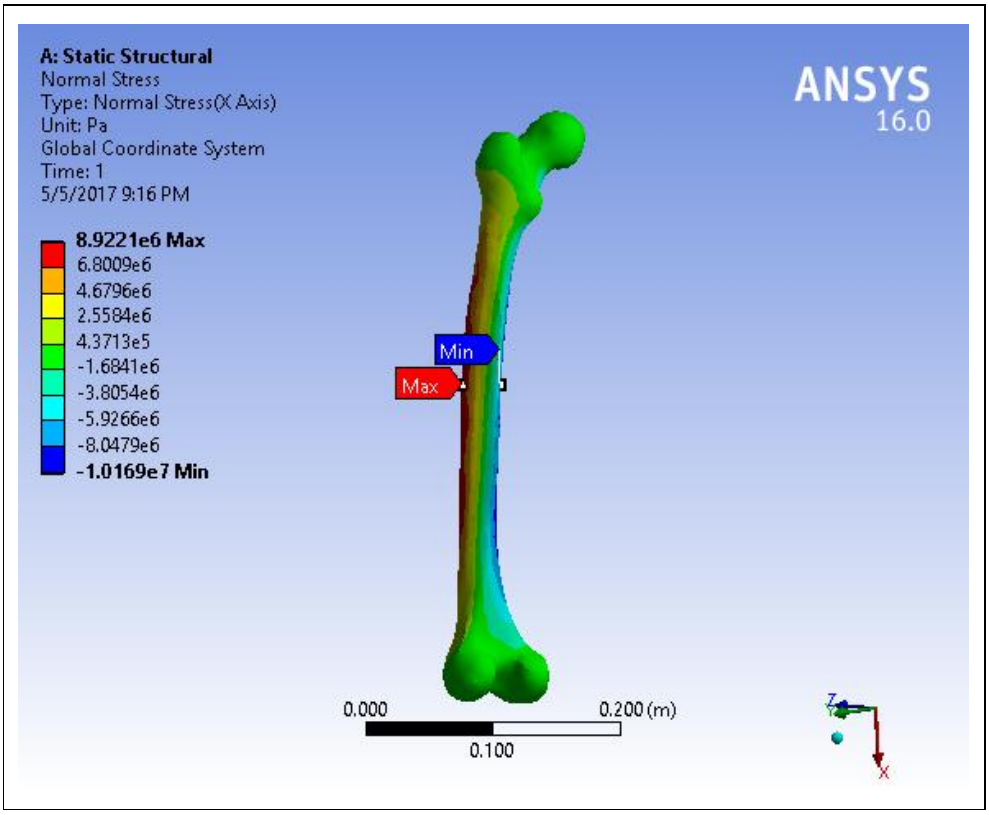


Figure 6-22: Normal Stress

comparison:

Table 6-3: Comparison Result between Experiment and literature

Max value	Experiment	Results of literature [4]	Error
Total deformation[m]	0.020143	0.023496	14.27 %
Principal stress [pa]	31760	30295	4.84 %
Equivalent (Von Mises) Stress [pa]	31717	31087	2.03 %
Fatigue Life	1×10^9	1×10^9	0 %
Safety factor	15	15	0 %

Table 6-4: Comparison Result between Experiment and literature

Max value	Experiment	Results of literature [20]	Error
Normal Stress [Mpa]	8.9221	6.9	29.31 %

6.6 Conclusion:

The differences of literature and experiment are associated with following factors:

a) the geometry of the model.
b) the orientation and the application of the hip contact force this factor is important because the dominant effect in normal walking is hip contact force (major effect).
c) muscles forces which are not included in my present study.
d) the material properties used for the model and the behavior of the bone.

6.7 Recommendation and Future work:

1. calculate and analyze the force on other bones of the body
2. the orientation and the application of the hip contact force this factor is important because the dominant effect in normal walking is hip contact force (major effect).

3. muscle forces this factor is important because the dominant effect in this activity is the muscle forces (Major effect) which are not included in my study.

References

[1] 3D systems, Geomagic User Guide, 3D systems, USA, 2013.

[2] M. Iqbal, A. Wasy, D. Batani, H. Rashid, and M. Lodhi, Design modification in rotor blade of turbo molecular pump, Nuclear Instruments and Methods in Physics Research Section A: Accelerators, Spectrometers, Detectors and Associated Equipment, 678(2012), pp.88-90.

[3] Structures, ANSYS, http://www.ansys.com/ (10/11/2016).

[4] R. Nareliya, K. Veerendra, Biomechanical Analysis of Human Femur Bone, IJEST, International Journal of Engineering Science and Technology, 3(2011), pp. 76-80.

[5] A. Latif Aghili, A.M. Goudarzi, M.H. Hojjati, S.M. Rabiee, Misagh Imani and Ali Paknahad, Biomechanical Investigation of Human Femur by Reverse Engineering as a Robust Method and Applied Simplifications, World Applied Sciences Journal, 28 (2013), pp. 2152-2157.

[6] Liming Voo, Mehran Armand, and Michael Kleinberger, Stress Fracture Risk Analysis of the Human Femur Based on Computational Biomechanics, JOHNS HOPKINS APL TECHNICAL DIGEST, 25(2004).

[7] K. Gok, B. Taspinar, F. Taspinar, E. Afsar, Arif Gok, M. Burak Bilgin, Research using finite element method of biomechanical behaviours of human femur model under the different loads, JEF, Journal of Engineering and Fundamentals, 1(2014), pp.43-47.

[8] M.T. Triband, F.W. Tillostson, and J.D. Martin, "Reverse and Re-Engineering in the DOD Organic Maintenance Community: Current Status and Future Direction," Technical Report 96-060, Applied Research Laboratory, the Pennsylvania State University, Feb. 1996.

[9] Broacha and Young : http://orthoinfo.aaos.org/topic.cfm (17/11/2016).

[10] J.C. Owen, Feature-Based Reverse Engineering, master's thesis, Dept. of Computer Science, University of Utah, 1994.

[11] 3D Point Cloud Segmentation: A survey, 2013 6th International Conference on Robotics, Automation and Mechatronics (RAM).

[12] H.J. de St. Germaine, S.R. Stark, W.B. Thompson, and T.C. Henderson, "Constraint Optimization and Feature-Based Model Construction for Reverse Engineering," Proceedings of the ARPA Image Understanding Workshop, Feb. 1996.

[13] C. Hsu, Graph-Based Approach for Solving Geometric Constraint Problems, doctoral dissertation, Dept. of Computer Science, University of Utah, 1996. 3D SOFTWARE - STUDIO 2012: Applied precision 3D digitizing system and services INC. 208 Britannia Rd. E., Unit 3, Canada L4Z126, pp. 1-2.

[14] Types of 3D Scanners and 3D Scanning Technologies, EMS, https://www.ems-usa.com/ (29/11/2016).

[15] Finite Element Method, NPTEL, http://nptel.ac.in/ (8/12/2016).

[16] G. P. Nikishkov, Introduction to the Finite Element Method, University of Aizu, Japan, 2004.

[17] The Femur, TeachMeAnatomy, http://teachmeanatomy.info/ , (3/12/2016)

[18] Joseph Hamill, Kathleen M. Knutzen, Timothy R. Derrick, Biomechanical Basis of Human Movement, Wolters Kluwer, Netherlands, 1995.

[19] Principles and Implant and instrumentation, ao foundation, https://www.aofoundation.org , (22/10/2016)

[20] A.E. Yousif, M.Y. Aziz, Biomechanical Analysis of the human femur bone during normal walking and standing up, IOSRJEN, International Organization of Scientific Research Journal of Engineering, 2(2012), pp.13_19.

Appendices

Appendix A

ANSYS Workbench