DEVELOPMENT OF IMPACT STRENGTH MODEL FOR HUMAN SYNTHETIC BONE

NG KOK CHUAN

BACHELOR OF ENGINEERING

UNIVERSITI MALAYSIA PAHANG

2010

NG KOK CHUAN BACHELOR OF MECHANICAL ENGINEERING 2010 UMP

DEVELOPMENT OF IMPACT STRENGTH MODEL FOR HUMAN SYNTHETIC BONE

NG KOK CHUAN

BACHELOR OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled "Development of Impact Strength Model for Human Synthetic Bone" is written by Ng Kok Chuan. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering.

Examiner Signature

DEVELOPMENT OF IMPACT STRENGTH MODEL FOR HUMAN SYNTHETIC BONE

NG KOK CHUAN

Thesis submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

DECEMBER 2010

SUPERVISOR’S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature

Name of Supervisor: Dr. DAW THET THET MON

Position: SENIOR LECTURER OF FACULTY OF MECHANICAL ENGINEERING

Date:

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurently submitted for award of other degree.

Signature
Name: NG KOK CHUAN
ID Number: MA07039
Date:

Dedicated to my parents

ACKNOWLEDGEMENT

I am grateful and would like to express my sincere gratitude to my supervisor Dr. DAW THET THET MON for her invaluable guidance and constant support in making this thesis. This thesis could not been done without her who not only served as my supervisor but also encourage me throughout the process. I also would like to express very special thanks to the panel member who give valuable comment and suggestion during my project presentation.

My sincere thanks go to all laboratory assistant from Faculty of Mechanical and Faculty of Manufacturing, Universiti Malaysia Pahang for their teaching and help during the period of my project.

I acknowledge my sincere indebtedness and gratitude to my parents, Mr. NG KIM LING and Mdm. SEOW YOK KUAN for their love and sacrifice throughout my life. They have always supported my dreams. They did a fine job raising me.

ABSTRACT

Due to current demand, the usage of synthetic bone had increased dramatically during these recent years. The main purpose of this project is to develop and analyze the impact strength model for human synthetic bone. The impact test simulation was done in finite element analysis (FEA). The bone model geometry will be designed in Solidwork environment and imported into finite element environment for FEA Finite element analysis will be carried out in MSC. Patran/Marc and Algor respectively. Major input data to FEM were isotropic material model, solid element type (tetrahedral in MSC. Marc and brick in Algor), constraints and impact load. Material properties were taken from literature. The model were validated with published experimental results. Predicted stress-strain response of synthetic bone was agreeable with published report. Based on the analysis results, MSC. Marc model is better than Algor. The result obtained shows the response of synthetic bone towards impact loading.

ABSTRAK

Dengan timbulnya permintaan semasa, permintaan terhadap penggunaan tulang sintetik meningkat secara mendadak sejak beberapa tahun yang lepas. Matlamat utama projek ini adalah untuk mengembangkan dan analisa kekuatan impak modal untuk tulang sintetik manusia. Simulasi ujian impak telah dijalankan dengan menggunakan FEA. Geometri tulang akan dibentuk dalam Solidwork dan diimport ke FEA. FEA akan dijalankan di MSC. Patran/Marc dan Algor. Data utama ke FEM adalah bahan modal isotropik, elemen pepejal (tertra dalam MSC. Marc dan bata dalam Algor, batas dan bban impak Sifat bahan diambil dari sastera. Model tersebut disahkan dengan keputusan experimen yang diterbitkan. Respon stress-tegang yang dijangka adalah dipersetujui dengan laporan yang diterbitkan. Berdasarkan keputusan analisa, modal MSC. Marc adalah lebih baik daripada modal Algor. Keputusan dari analisa menunjukkan penbalasan untuk tulang sintetik manusia terhadap impak.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

L	Lagrangian
T	Kinetic energy
π	Potential energy
m	Mass
v_o	Velocity
E	Modulus of elasticity
V	Volume
P_m	Static force
L	Length
I	Moment of inertia
U_m	Strain energy
C	Radius
V_e	Element volume

LIST OF ABBREVIATIONS

FEA	Finite element analysis
TCP	Tricalcium phosphate
CAE	Computer Aided Engineering
CAD	Computer Aided Design
MSC	MacNeal-Schwendler Corporation
FEM	Finite element method

REFERENCES

Gladius Lewis, Scott Mladsi, 1999. Correlation between impact strength and fracture toughness of PMMA-based bone cements. *Biomaterials 21 (2000) 775-781*

John M. Dawson *, Boris V. Khmelniker, Mark P. McAndrew, 1998. Analysis of the structural behavior of the pelvis during lateral impact using the finite element method. *Accident Analysis and Prevention 31 (1999) 109–119*

B.P. Kneubuehl a,*, M.J. Thalib, 2003. The evaluation of a synthetic long bone structure as a substitute for human tissue in gunshot experiments. *Forensic Science International 138 (2003) 44–49*

F.A. Bandak *, R.E. Tannous, T. Toridis, 1999. On the development of an osseo-ligamentous ®nite element model of the human ankle joint. *International Journal of Solids and Structures 38 (2001) 1681±1697*

Steele, D. Gentry. & Bramblett, Claud A. 1988 *The anatomy and biology of the human skeleton / D. Gentry Steele, Claud A. Bramblett ; photographs by Virginia K. Massey, Jean M. Christiansen, and D. Gentry Steele* Texas A&M University Press, College Station

Mechanics of Materials 4th edition in SI units Ferdinand P. Beer

Hall, Susan. *Basic Biomechanics*. Fifth Edition. Pg. 88

Mechanical Behaviour of Materials 2th edition Marc Meyers and Krishan Chawla

Introduction to Finite Elements in Engineering 3rd edition Tirupathi R. Chandrupatla and Ashok D. Belegundu

PATRAN
Retrieved from http://www.mscsoftware.com/Products/CAE-Tools/Patran.aspx

TCP
Retrieved from http://www.kasios.com/doc-pdf/TCP-891-eu.pdf

APPENDIX A
SAMPLE CALCULATION FOR STRESS AND STRAIN

The strain energy for the strain energy of the cantilever beam is

$$U_m = \frac{{P_m}^2 L^3}{6EI}$$

Where P_m = static force L = length

E = modulus of elasticity I = moment of inertia

$$I = \frac{1}{4}\pi\,(0.004)^4 = 2.0106\;x\;10^{-10}\;m^4$$

$$U_m = 0.28J$$

$$P_m = \sqrt{\frac{(0.28)6(4x10^8)(2.0109x10^{-10})}{(0.02)^3}} = 129.958N$$

The maximum stress at at the fixed end B is

$$\sigma_m = \sqrt{\frac{6U_m E}{L(I/c^2)}}$$

Where U_m = strain energy E = modulus of elasticity

L = length I = moment of inertia

C = radius

$$\sigma_m = \sqrt{\frac{6\;(0.28)(4x10^8)}{(0.02)(2.0106x10^{-10}/0.004^2)}} = 51.709x10^6\;Pa$$

Strain tensor calculated is

$$E = \frac{\sigma}{\epsilon}$$

Where E = modulus of elasticity
σ = stress
ϵ = strain

$$\epsilon = \frac{51.709x10^6}{4x10^8} = 0.129$$

APPENDIX B
DATA VALUE FOR STRESS-STRAIN GRAPH

Time (s)	Stress (MPa)	Strain
2e-5	15.6	3.04e-3
4e-5	33.4	6.52e-3
6e-5	53.0	1.03e-2
8e-5	74.7	1.46e-2
1e-4	97.6	1.9e-2

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Almost all livings thing such as animals & humans have bones. Bones are rigid organs that form part of the skeleton. Bones can produces red and white blood cells and store minerals. Bones come in a variety of shapes to fit different area of the body as it provides a frame to keep the body supported. Bone is light in weight yet strong and hard. There are 206 bones in a human adult body and 270 in a human infant body. Bones serves as protection for internal organs, such as the skull protecting the brain. Bones combine and function together to generate and transfer forces so the human body parts can move in three-dimensional space. Daily activities pose danger to the bones where fracture might occurs due to various type of loading or force particularly heavier action and more extreme activities in current society increase the chances for bone fracture (The anatomy and biology of the human skeleton).

Artificial bone has been developed using bone-like material. It has now played an important role in bone graft procedures which replace human bone that was lost due to fracture. There are many researches done to develop one similar to natural bone in effort to maintain its functions in the human body. For example, artificial bone has porous structure which allows the growth of blood vessels. Bone properties are important to withstand different kinds of load especially impact load which is the main cause of fracture. Impact loading is the dynamic effect on a body and a forcible momentary contact of another moving body. The bone, however protected is very vulnerable to impact load. In this regard, knowledge of bone respond to impact load would be useful to improve bone properties. Unfortunately in literature, only limited study has been done to find out bone respond and these are typical tensile or compression test because of it being an expensive research and difficult to obtain natural bone test specimen.

In this project, the synthetic bone will be tested to develop impact strength model since it has similar properties to natural bone. Thus, it can show similar test result and respond for natural bone. This synthetic bone is commonly used in bone grafting operation for human. Finite element analysis is carried out to simulate the response of synthetic bone to impact load.

1.2 PROBLEM STATEMENT

Human are highly exposed to danger that might cause bone fracture especially from accidents. Accidents can be in form of car crash, fall from higher place, and extreme sports. As such, demands of bone grafting a process of replacing artificial bone have been increased. . There are many efforts to create artificial bone which is similar to natural bone or even a more improved bone property. An important property of artificial bone is the ability to withstand high impact loading. However the strength of artificial bone to withstand high impact loading is relatively unknown. The experimental study alone is expensive due to high cost of the specimen.

1.3 OBJECTIVES

Objectives for this project refer to the mission, purpose, or standard that can be reasonably achieved within the expected timeframe and with the available resources. The objectives of this project are:

1. To develop the finite element model that can predict impact strength of human synthetic bone.
2. To validate finite element model with experiment.
3. To investigate predicted stress-strain responds of synthetic bone under impact load.

1.4 PROJECT SCOPES

The scopes for this project are:

1. Bone geometry will be constructed using Solidwork.
2. Finite element model will be developed in MSC. Patran and Algor.
3. Impact analysis will be carried out in MSC. Marc and Algor.
4. Finite element model will be validated with published experimental result.
5. Stress-strain relationship due to impact load will be plotted in Excel.

Bone model size is scaled down to 12.5% of the actual one to resolve the meshing problem in finite element model. Two finite element codes are tested to compare its predictability. Solidworks model is imported to finite element environment for finite element modeling in MSC. Marc and Algor.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter will provide the detail description literature review done according to title of development of impact strength model for human synthetic bone. Literature regarding any development or experiment about bone properties is useful in this project. This is includes the bone sizes or shapes available at current market, synthetic bone material, type of test such as compression and impact, and finite element software available for analysis.

2.2 NATURAL BONE

The primary tissue of bone, osseous tissue, is a relatively hard and lightweight composite material, formed mostly of calcium phosphate in the chemical arrangement termed calcium hydroxylapatite (this is the osseous tissue that gives bones their rigidity). It has relatively high compressive strength, of about 1800 kg/cm^2 but poor tensile strength of 104-121 MPa, meaning it resists pushing forces well, but not pulling forces. While bone is essentially brittle, it does have a significant degree of elasticity, contributed chiefly by collagen. All bones consist of living and dead cells embedded in the mineralized organic matrix that makes up the osseous tissue. Bone is not a uniformly solid material, but rather has some spaces between its hard elements. The hard outer layer of bones is composed of compact bone tissue, so-called due to its minimal gaps and spaces. Its porosity is 5-30%. This tissue gives bones their smooth, white, and solid appearance, and accounts for 80% of the total bone mass of an adult skeleton. Compact bone may also be referred to as dense bone. Filling the interior of the bone is the trabecular bone tissue, which is composed of a network of rod- and plate-like elements that make the overall organ lighter and allow room for blood vessels and marrow. Trabecular bone accounts for the remaining 20% of total bone mass but has nearly ten times the surface area of compact bone. Its porosity is 30-90%. If for any reason there is an alteration in the strain to which the cancellous is subjected, there is a rearrangement of the trabeculae. The microscopic difference between compact and cancellous bone is that compact bone consists of haversian sites and osteons, while cancellous bones do not. Also, bone surrounds blood in the compact bone, while blood surrounds bone in the cancellous bone. Figure 2.1 shows section through the head of the femur, showing the cortex, the red bone marrow and a spot of yellow bone marrow. The white bar represents 1 centimeter. Specimen obtained after total hip replacement surgery, left hip (Basic Biomechanics).

Figure 2.1: Bone cross-section

Source: Wikimedia Commons

2.3 SYNTHETIC BONE DESIGN

There are many shapes and sizes of synthetic bone design. Available shapes can be granules, sticks, block, cylinder, and wedge. Sizes can range from 2mm to 14 mm for regular geometry. Different shapes and sizes serve different purpose or area of substitute in human bone. Figure 2.2 illustrates synthetic bone size and shapes available in the market today The synthetic bone is design with fully interconnected pores, having pore size between 200 to 500um, and mean porosity between 60 to 80 percent to resemble the real bone.

	Ref.	Shapes	Sizes
	K43105G-E	Granules	2-3mm [5cc]
	K43110G-E	Granules	2-3mm [10cc]
	K43115G-E	Granules	2-3mm [15cc]
	K43120G-E	Granules	2-3mm [20cc]
	K43105S-E	Sticks	5x5x20mm (box of 5)
	K43115B-E	Block	15x15x20mm (box of 1)
	K43120B-E	Block	15x20x30mm (box of 1)
	K43106W-E	Wedge	6mm (box of 1)
	K43108W-E	Wedge	8mm (box of 1)
	K43110W-E	Wedge	10mm (box of 1)
	K43112W-E	Wedge	12mm (box of 1)
	K43114W-E	Wedge	14mm (box of 1)

Figure 2.2: Synthetic bone shapes and sizes.

Source: KasiosTCP 2010

2.4 SYNTHETIC BONE MATERIAL

A synthetic bone, so-called TCP as trade name is made of pure β-TCP which is calcium phosphate molecule similar to the mineral phase of the natural bone. Mode of operation in curing bone defects is highly bioactive. It undergoes total or partial resorption and is replace by neoformed natural bone. It is indicated for filling bone voids or defects of the skeletal system (such as the extremities, spine and the pelvis) that are not intrinsic to the stability of the bony structure. These defects may be surgically created osseous defects or osseous defects created from traumatic injury to the bone. Kasios TCP is a bone graft substitute that resorbs and is replaced with bone during the healing process.

It has many features and benefits including no reaction to foreign body, can be replaced by natural bone, encourages a quick and proper osseointegration, no risk of immune response, no risk of cross contamination, no risk of disease transmission, decrease surgery time, allows for a long term follow up, and fill irregulary shaped cavities completely (Kasios TCP 2010).

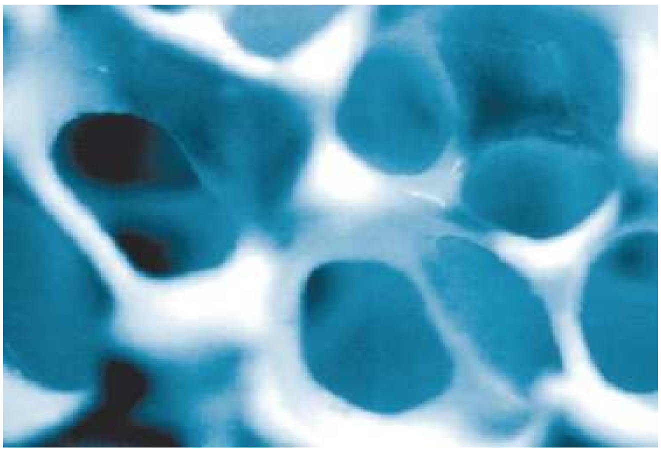

Figure 2.3: Synthetic bone microscopic view.

Source: Kasios TCP 2010

2.5 IMPACT LOAD

In mechanics, an impact is a high force or shock applied over a short time period when two or more bodies collide. Such a force or acceleration usually has a greater effect than a lower force applied over a proportionally longer time period of time. The effect depends critically on the relative velocity of the bodies to one another.

2.6 LAB BASED TEST

Impact in laboratory scale is testing an object's ability to resist high-rate loading. Alternatively, it is a test for determining the energy absorbed in fracturing a test piece at high velocity. Most of us think of it as one object striking another object at a relatively high speed. There are basically two types of impact tests: pendulum and drop weight depending on how the load is applied. It can be further classified as Izod, Charpy, and tensile impact from view point of specimen geometry and size. (Instron 2010).

2.6.1 Pendulum Testing

The first attempts at obtaining this value were done by means of a swing pendulum. A pendulum of a known weight is hoisted to a known height on the opposite side of a pivot point. By calculating the acceleration due to gravity (32.2 ft/sec2 or 9.8 m/sec2), the engineer knows that the weight falling from a set height will contain a certain amount of impact energy at the bottom of the swing. By clamping or supporting a specimen on the bottom, the sample can be released to strike and break the specimen. The pendulum will continue to swing up after the break event to a height somewhat lower than that of a free swing. The engineer can use this lower final height point to calculate the energy that was lost in breaking the specimen. Many pendulum machines will incorporate a pointer and energy reading device so that calculation is unnecessary. Examples of pendulum testing are Charpy and Izod testing (Instron 2010). Figure 2.4 shows a typical Izod impact testing machine.

Figure 2.4: Analog Charpy Izod Impact Testing Machine FIT 300

Source: Fine Manufacturing Industries 2010

2.6.2 Drop Weight Impact Test

A second method was to drop a weight in a vertical direction, with a tube or rails to guide it during the "free fall." With the height and weight known, impact energy can be calculated. In the early days, there was no way to measure impact velocity, so engineers had to assume no friction in the guide mechanism. Since the falling weight either stopped dead on the test specimen, or destroyed it completely in passing through, the only results that could be obtained were of a pass/fail nature.

Falling weight impact has several key advantages over other methods.

1. It is applicable for molded samples, molded parts, etc.
2. It is unidirectional with no preferential direction of failure. Failures originate at the weakest point in the sample and propagate from there.
3. Samples don't have to shatter to be considered failures. Failure can be defined by deformation, crack initiation, or complete fracture, depending on the requirements.

These factors make falling weight testing a better simulation of functional impact exposures, and therefore closer to real-life conditions. However, there are drawbacks to uninstrumented falling weight and Gardener or Gardner weight drop testing (Instron 2010).

Figure 2.5: Gardner Impact Tester

Source: Qualitest 2010

2.7 FINITE ELEMENT THEORY

In dynamic analysis, lagrangian, L is defined by

$$L = T - \pi \tag{2.1}$$

where T is the kinetic energy and π is the potential energy

`L also can be expressed in terms of the generalized variables (q_1, q_2,…, q_n, $\dot{q}_1, \dot{q}_2, \ldots, \dot{q}_n$) where

$$\frac{d}{dt}\left(\frac{\partial L}{\partial \dot{q}_i}\right) - \frac{\partial L}{\partial q_i} = 0 \qquad i = 1\ to\ n \tag{2.2}$$

For solid body with distributed mass, as illustrated in Figure 2.6,

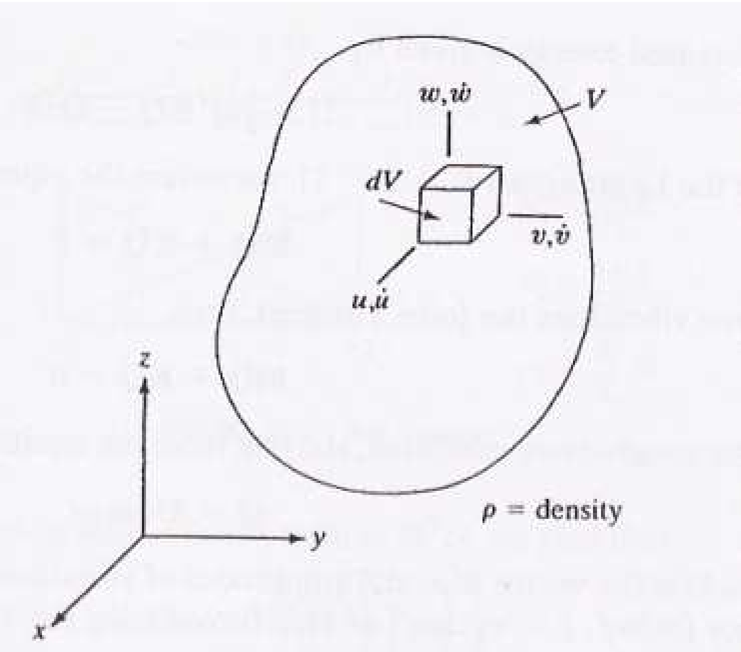

Figure 2.6: Body with distributed mass

Source: Introduction to finite element in engineering

The density of the material is p and the velocity vector of the point x_1 is

$$\dot{u} = [\dot{u}, \dot{v}, \dot{w}]^{t} \tag{2.3}$$

Each element can be expressed as u in terms of nodal displacement **q** and shape function **N**.

$$u = Nq \tag{2.4}$$

However in dynamic analysis, element of **q** are dependent on time while **N** represents shape function defined on a master element. Thus, velocity vector is

$$\dot{u} = N\dot{q} \tag{2.5}$$

By substituting the above equation into kinetic energy equation

$$T_e = \frac{1}{2}\dot{q}^T \left[\int_e pN^T N \, dV\right] \dot{q} \tag{2.6}$$

Where the bracketed expression is the element mass matrix

$$m^e = \int_e pN^T N \, dV \tag{2.7}$$

This mass matrix is called the consistent mass matrix. By summing all the element,

$$T = \sum_e T_e = \sum_e \frac{1}{2}\dot{q}^T m^e q = \frac{1}{2}\dot{Q}^T M Q \tag{2.8}$$

The potential energy is

$$\pi = \frac{1}{2}Q^T K Q - Q^T F \tag{2.9}$$

Then from L = T – π, the equation of motion in finite element analysis can be generalized as

$$M\ddot{Q} + KQ = F \tag{2.10}$$

Where M is assembled mass matrix, K is assembled stiffness matrix.

2.8 FINITE ELEMENT SOFTWARE

Nowdays, variety of finite element software are are available to solve conveniently finite element equation for which manual solving cannot be used. Some popular finite element codes are MSC, Algor, ABAQUS, ANSYS.

Patran is a widely used pre/post-processing software for Finite Element Analysis (FEA), providing solid modeling, meshing, and analysis setup for MSC Nastran, Marc, Abaqus, LS-DYNA, ANSYS, and Pam-Crash (MSC software 2010).

Designers, engineers, and CAE analysts tasked with creating and analyzing virtual prototypes are faced with a number of tedious, time-wasting tasks. These include CAD geometry translation, geometry cleanup, manual meshing processes, assembly connection definition, and editing of input decks to setup jobs for analysis by various solvers. Pre-processing is still widely considered the most time consuming aspect of CAE, consuming up to 60% of users' time. Assembling results into reports that can be shared with colleagues and managers is also still a very labor intensive, tedious activity (MSC software 2010)

2.9 ANALYIS OF BONE USING FINITE ELEMENT METHOD

Often simple shapes and regular load configurations are prerequisite to attaining an analytical solution to a problem in structural mechanics. The finite element method is one of several numerical approaches to a solution. Its use enables engineers to solve practical problems which have realistic shapes and complex loads. The solid structure is represented as an assembly of a finite number of elemental volumes. Each volume, or element, represents a real space of material within the structure. Highly complicated geometries can be easily modeled as an assembly of simply-shaped, prismatic elements. A finite element analysis is often performed to provide preliminary estimates of deformation, strain and stress in a complicated structure. Several finite element models with the same geometry but with increasing numbers of elements and nodes were constructed to demonstrate convergence of the solution. One half to two-thirds of pelvic fractures are caused by motor-vehicle accidents. A study of accident victims showed that 7.5% of frontal impacts produce pelvic fractures but among those killed in lateral impacts, more than 50% had a pelvic fracture (Gocken et al., 1994). One third of survivors of pelvic fractures have unsatisfactory outcomes after treatment (Dalal et al., 1989; Burgess et al., 1990; Pattimore et al., 1992). These njuries present challenging orthopaedic problems and their prevention is an important aspect of automobile design (Accident Analysis and Prevention 31 (1999) 109–119).

For instance, in regions where stresses of large magnitude were expected (such as the auricular surfaces and the ilio- and ischio-pubis), the size of the elements was reduced. To predict local bone failure, a Von Mises stress yield criteria was used for the pelvic bone. Von Mises stress was chosen because of its convenience to represent general stress distribution within the structure.According to this theory of yielding, failure occurs when the energy of distortion per unit volume is equal to that associated with yield in a simple tension test (Accident Analysis and Prevention 31 (1999) 109–119).

2.10 IMPACT THEORY

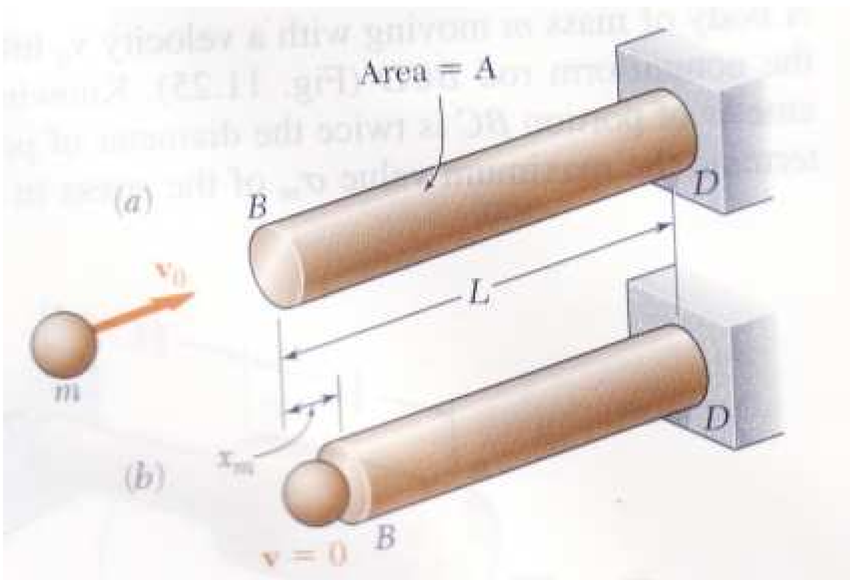

Figure 2.7: Impact load

Source: Mechanics of Materials 4th edition

From Figure 2.7, consider a rod BD of uniform cross section which is hit at its end B by a body of mass m moving with a velocity v_o. As the rod deforms under the impact, stresses develop within the rod and reach a maximum value of stress. After vibrating for a while, the rod will come to rest, and all stresses will disappear. Such a sequence of events is referred to as an impact loading (Ferinand P. Beer 2006).

Strain energy corresponding to the maximum deformation Xm is

$$U_m = \frac{1}{2} m {v_o}^2 \tag{2.11}$$

Where m = mass v_o = velocity

This equation is valid with the following assumptions:

1. No energy should be dissipated during the impact.
2. The striking body should not bounce off the structure and retain part of its energy.
3. Kinetic energy of striking body is transferred entirely to the structure.

Then, for impact load shown in Figure 2.7, maximum stress was derived as

$$\sigma_m = \sqrt{\frac{2U_m E}{V}} = \sqrt{\frac{m {v_o}^2 E}{V}} \tag{2.12}$$

Where

m = mass E = modulus of elesticity

v_o = velocity V = volume

On the other hand, for impact loading shown in Figure 2.8, the strain energy for the strain energy of the cantilever beam is

$$U_m = \frac{{P_m}^2 L^3}{6EI} \tag{2.13}$$

Where P_m = static force L = length

E = modulus of elasticity I = moment of inertia

The maximum stress at at the fixed end B is

$$\sigma_m = \sqrt{\frac{6U_mE}{L(I/c^2)}} \tag{2.14}$$

Where U_m = strain energy E = modulus of elasticity

L = length I = moment of inertia

C = radius

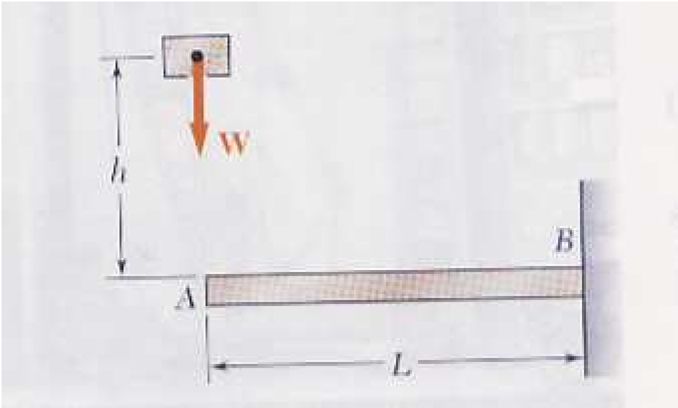

Figure 2.8: Load applied on free end of cantilever beam

Source: Mechanics of Materials 4th edition

2.11 MECHANICAL PROPERTIES

Material properties are vital in order to obtain an accurate result. Figure 2.9 shows the range of elastic moduli of biological cells and conventional materials.

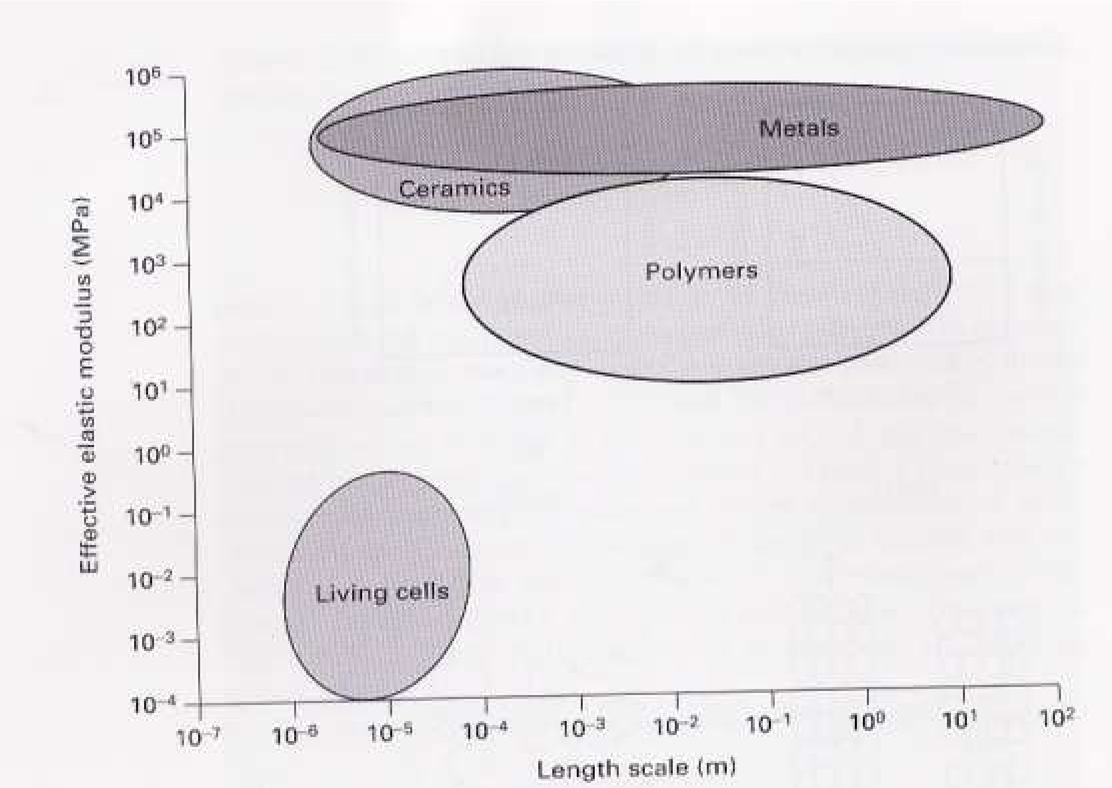

Figure 2.9: Elastic moduli of biological cells and conventional materials

Source: Mechanical Behaviour of Materials 2th edition

Biological materials have complex elastic properties. Soft tissues exhibit nonlinear elasticity. Hard tissues, such as bone, have a linear elastic response conditioned by their density.

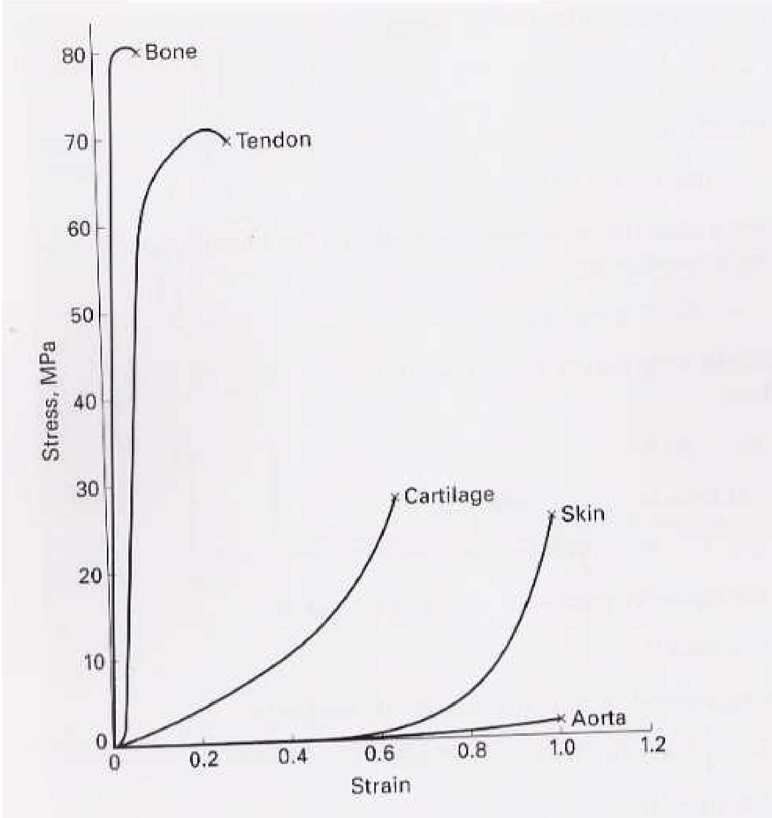

Figure 2.10: Stress-strain response for biological materials

Source: Mechanical Behaviour of Materials 2th edition

Figure 2.10 shows the stress-strain response of a number of biological materials. The strength increases as the ductility decreases for the case with synthetic material. The strongest material in our body is the cortical bone. For our project, we will focus on the mechanical performance of the bone. There are two principle types of bone: cortical (or compact) and cancellous (or porous) Figure 2.11 provides important mechanical properties for a number of biological materials.

Table 3.4 Mechanical Properties of Some Biological Materials

Material	E (MPa)	Fracture Stress (MPa)	Strain at Fracture
Elastin	0.6		
Resilin	1.8		
Collagen	1,000	70	0.09
Fibroin	10,000		
Cortical bone –			
Longitudinal	$(14–24) \times 10^3$	150	~0.015
Transverse	$(8–18) \times 10^3$	50	
Cancellous (porous) bone	10–200		
Cellulose	80,000	1000	0.024
Tendon	1,300	75	0.09
Keratin	2,500	50	0.02
Alpha (mammalian) Beta (birds)	2,000	20	
Dentine		300	
Spider Silk (radial)		1,500	0.06
Silkworm Silk		500	

Figure 2.11: Mechanical properties for biological materials

Source: Mechanical Behaviour of Materials 2^{th} edition

2.12 STRESS-STRAIN CURVE

At a microscopic level, deformation in polymers involves stretching and rotating of molecular bonds. The deformation mechanisms in polymers can be divided into brittle, ductile, and elastomeric. Polymers are viscoelastic, which means their stress-strain behavior is dependent on time. Bone shows deformation like a brittle mechanism. Figure 2.12 shows the types of stress-strain curves in a polymer.

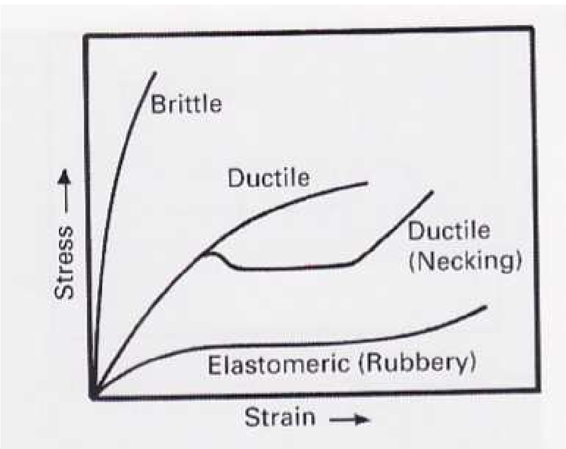

Figure 2.12: Stress-strain of polymer.

Source: Mechanical Behaviour of Materials 2^{th} edition

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter will provide the detail explanation on the methodology of this project “Development of impact strength model for human synthetic bone” from the beginning till the end. Methodology can properly refer to the theoretical analysis of the methods appropriate to a field of study or to the body of methods and principles particular to a branch of knowledge. The methodology act as the guidance or step that needs to be follow and this will ensure the project done according to the planning. Methodology works as an algorithm that finds a solution in the given environment of the multi-layered finite space consisting of literature review. This includes identifying the suitable material and design, creating a Solidwork model of the design, analysis the model, and documentation.

In analysis with MSC. Marc, dynamic load is applied directly to the bone and non-linearity was taken into account for large displacement. Whereas in Algor, interactive effect between the bone and the pendulum was analyzed. Velocity and boundary condition was acquired to pendulum.

3.2 FLOW CHART

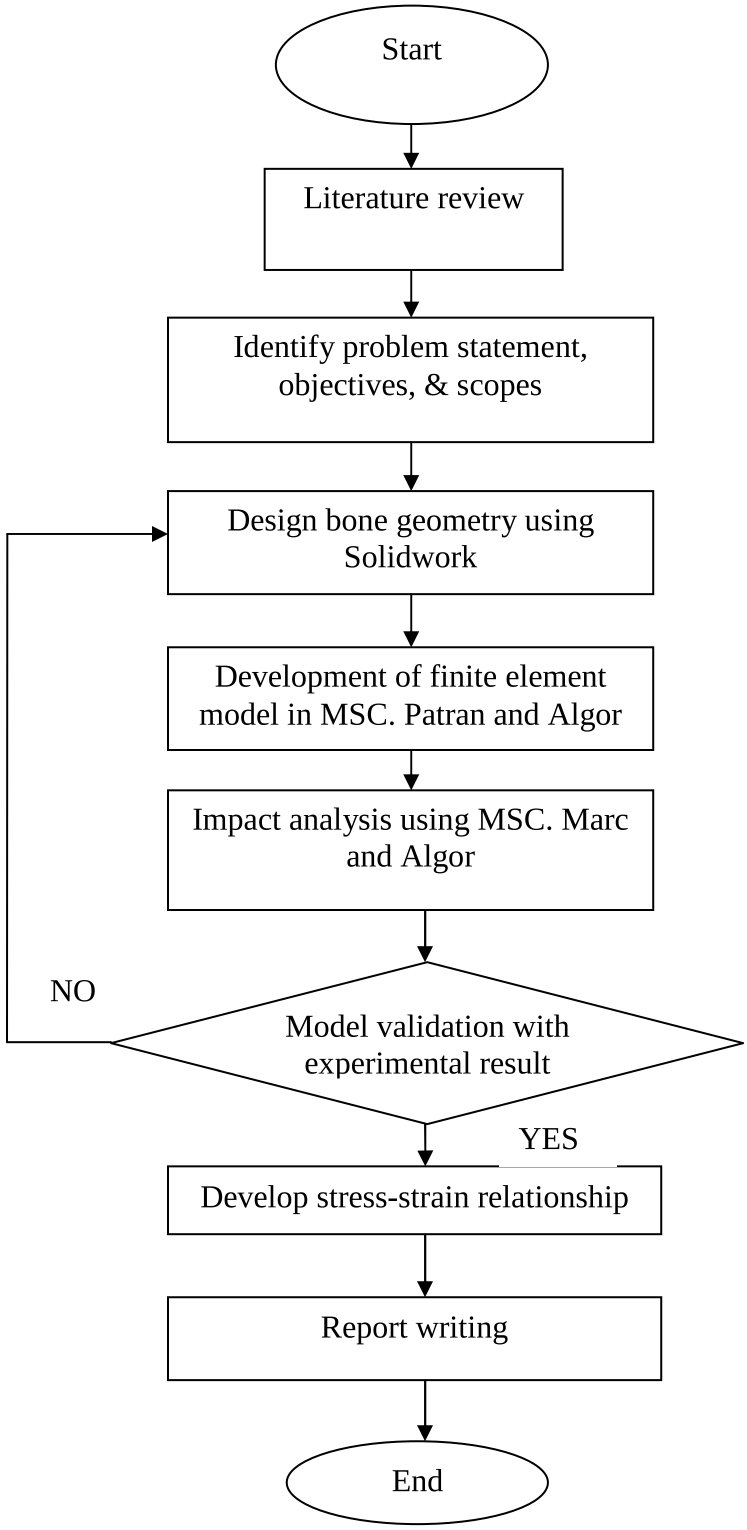

Figure 3.1: Flow chart

Figure 3.1 shows the project flow which provides a better understanding on this project. Several researches on internet and market available product information is useful to provide better understanding on the project layout. Literature review is vital in getting a rough idea on project which can be set as a foundation.

In the following steps, problem statement, objectives, and scopes were identified. This provides a guideline of procedures in order to complete the project. It also functions as a checklist of things to be done by the end of the project.

Synthetic bone model is designed using Solidworks. Porosity of same size in the actual ibe was created to make the model realistic. The final shape is a cylindrical model.

Solidwork model was imported to finite element environment in MSC. Patran and Algor to develop finite element model. Boundary condition and element model properties are defined.

The solver used in MSC. Patran is MSC. Marc. After element properties and boundary condition is loaded, the analysis is carried out. The model is validated with experimental result. The procedure is repeated from designing a human synthetic bone model using Solidworks if there was a significant difference in the result

Once the result is validated with the experimental result, stress-strain relationship respond of the synthetic bone is being developed. The final stage of the project process is documentation. Documentation is the full report which consists of introduction, literature review, methodology, result and discussion and conclusion.

3.3 BONE GEOMETRY

The design criteria are based on the shape, size, & similarity in resemblance with the current market human synthetic bone. For this case, a cylindrical shape with standard dimension was design to project better resemblance with real result. Using Solidworks, several designs was made before the final design.

3.3.1 Design 1

Solidwork is started and a new file is created. A plane is chosen and a rectangular of 2mm by 4mm is drawn on the selected plane. Then, the rectangle is extruded by 6mm to form a rectangular block. It has a final dimension of 2mm x 4mm x 6mm. The porosity was created by repeat extruding cut from two top side of the design along the 6mm with 45 degree from opposite side which intersects one another.

3.3.2 Design 2

From Design 1, a plane was created on the surface of the rectangular block at the middle. A circle of 1mm radius is drawn at the newly created plane. Then a cylindrical shape is created by reverse extrude the circle by 5mm along the 6mm length of the rectangular block. Design 2 has a dimension of π x 1mm x 1mm x 5mm. This design is a 25 percent scale down of the real human synthetic bone dimension of π x 4mm x 4mm x 20mm.

3.3.3 Design 3

From Design 2, a smaller circle is drawn with a radius of 0.5mm on the same plane created earlier. Then a smaller cylindrical shape is created by reverse extrude the circle by 2.5mm along the 5mm length of the previous cylindrical block. Design 3 has a dimension of π x 0.5mm x 0.5mm x 2.5mm. This design was further reduce to half the dimension of Design 2 which is a 12.5 percent scale down of the real human synthetic bone.

3.4 MODEL CONSIDERATION

Two finite elements analysis software was used for this project which is MSC. Marc and Algor. For MSC. Marc, only bone model geometry will be imported. For Algor, both the bone and pendulum model geometry will be imported. In both cases, the bone and pendulum model geometry is scaled down to 12.5% of the real dimension.

3.5 FINITE ELEMENT ANALYSIS

Two finite element analysis software was involved which is Algor and MSC. Patran/Marc was used to develop and analyze the impact model. The differences in the both software will be compared. Figure 3.2 shows the basic input data needed for finite element analysis.

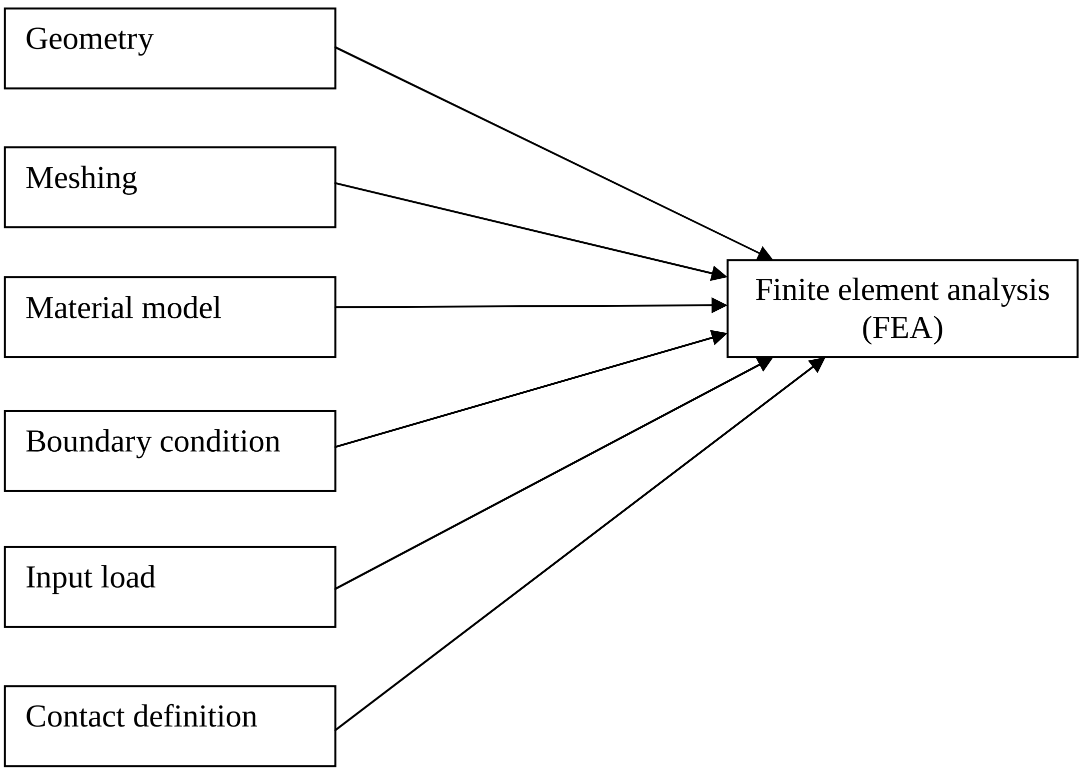

Figure 3.2: Input to FEA

Model geometry used for impact analysis is 3D solid. The model was constructed in Solidworks. The Solidwork model was saved as Parasolid format for MSC. Patran and IGES format for Algor.

Element is an important property in finite element analysis. In this analysis, tetrahedral element type was used. For MSC. Patran, topology of Tet4 is used to mesh the bone model geometry which produced a number of 35968 elements based on mesh value of 0.05. For Algor, solid meshing with tetrahedral element is used. A value of 120% in meshing produces a number of 18091 elements.

Each part of model geometry must be assigned with at least one specific material. Material type and value is important to obtain a better result. For MSC. Marc material model, approximate elastic properties were justified based on Figure 2.10, where no apparent plasticity exist in bone failure. Hence, the material model was described in Figure 3.3.

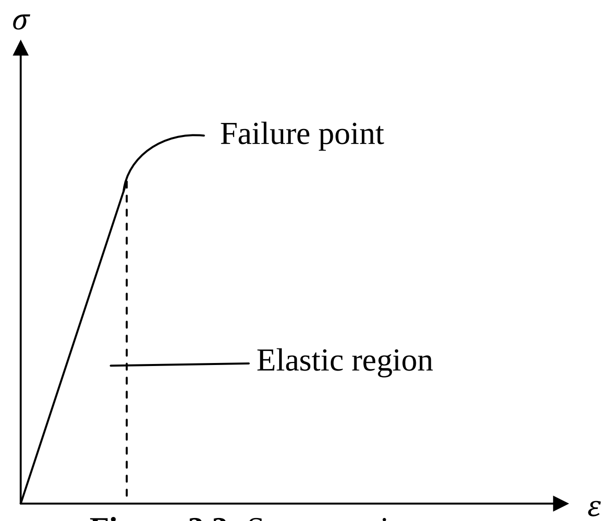

Figure 3.3: Stress-strain curve

On the other hand, Drucker-Prager was incorporated in Algor. The Drucker-Prager material model is available for 2-D, brick and tetrahedral elements. This model is similar to the elastic/plastic material model described above except that the assumption of a yielding function is different. Basically, this material model assumes that the volumetric strain changes the yielding function, which occurs in granular materials such as concrete and rock. The Drucker-Prager material properties are listed below. In addition it may be necessary to define some isotropic material properties. The 3-D Drucker-Prager material model is used to model geological materials, such as soils, clays and rocks. This material model uses a yield function defined by

$$F(t) = \alpha I_1(t) + \sigma(t) - \beta \tag{3.1}$$

Where

$$I_1(t) = \sigma_{11}(t) + \sigma_{22}(t) + \sigma_{33}(t) \tag{3.2}$$

$$\sigma(t) = \sqrt{\frac{1}{2}\sum_{i,j=1}^{3} S_{ij}(t)S_{ij}(t)} \tag{3.3}$$

$$S_{ij}(t) = \sigma_{ij} - \frac{1}{3}\delta_{ij}I_1(t) \tag{3.4}$$

Figure 3.4 shows the Drucker-Prager yield function

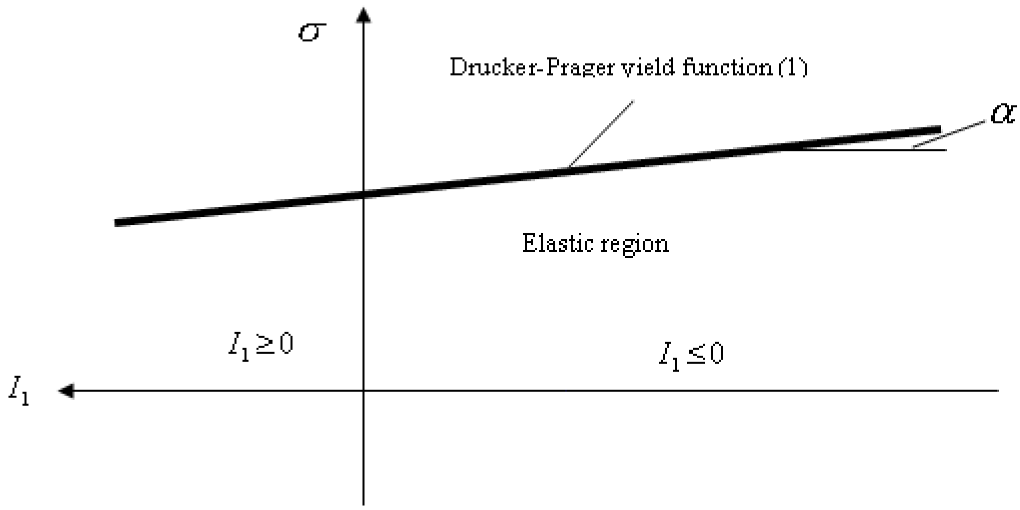

Figure 3.4: Graph of the Drucker-Prager yield function.

Table 3.1 shows the value used for both of the finite element analysis

Table 3.1: Material properties input value

Software	MSC. Patran	Algor
Young Modulus	4000	400
Poisson ratio	0.17	0.1
Density	1.6e-9	1.6e-9
Alpha		3.7
Beta		80

Boundary condition is constraints which applied in finite element analysis to restrict the movement of the node or surface. The constraint type is translation, rotation, and contact.

Input load controls the analysis parameters and the output of the analysis result. Type of result can be control at the output request. Input load also control the analysis solver as well as the duration of the analysis. Impact load was designed by controlling model displacement and time in MSC. Marc whereas in Algor, velocity designed to pendulum. Other input data required in each model analysis was summarized in Table 3.2.

Table 3.2: Analysis input data

Input	MSC. Marc	Algor
Analysis type	Non-linear Static with large displacement	Mechanical Event Simulator (MES)
Contact	No	Yes (Surface to surface)
Adaptive	Yes	No

Model validation was done quantitatively based on absorbed energy from impact load. Stress-strain values were exported to Excel to develop stress-strain curve.

3.6 FINITE ELEMENT FORMULATION

The kinetic energy and the potential energy are derived as

$$T = \frac{1}{2}\dot{Q}^T M Q \tag{3.5}$$

$$\pi = \frac{1}{2} Q^T K Q - Q^T F \tag{3.6}$$

Being Q and $\dot{Q}$ are displacement and velocity terms respectively

Using L = T – π, the equation of motion is

$$M\ddot{Q} + KQ = F \tag{3.7}$$

Considering material density p to be constant, the element mass matrices is

$$m^e = p \int_e N^T N \, dV \tag{3.8}$$

For tetrahedral element

$$N = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 & 0 & 0 \\ 0 & N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 & 0 \\ 0 & 0 & N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 \end{bmatrix} \tag{3.9}$$

From stress-strain relation, the assumed displacement field $u = Nq$ will yield $\epsilon = Bq$ where B is constant matrix expressed as

$$B = \begin{bmatrix} A_{11} & 0 & 0 & A_{12} & 0 & 0 & A_{13} & 0 & 0 & -\check{A}_1 & 0 & 0 \\ 0 & A_{21} & 0 & 0 & A_{22} & 0 & 0 & A_{23} & 0 & 0 & -\check{A}_2 & 0 \\ 0 & 0 & A_{31} & 0 & 0 & A_{32} & 0 & 0 & A_{33} & 0 & 0 & -\check{A}_3 \\ 0 & A_{31} & A_{21} & 0 & A_{32} & A_{11} & 0 & A_{33} & A_{23} & 0 & -\check{A}_3 & -\check{A}_2 \\ A_{31} & 0 & A_{11} & A_{32} & 0 & A_{12} & A_{33} & 0 & A_{13} & -\dot{A}_3 & 0 & -\check{A}_1 \\ A_{21} & A_{11} & 0 & A_{22} & A_{12} & 0 & A_{23} & A_{13} & 0 & -\check{A}_2 & -\check{A}_1 & 0 \end{bmatrix} \tag{3.10}$$

Where the element stiffness matrix k^e is

$$k^e = V_e B^T D B \tag{3.11}$$

Where,

$$D = \frac{E}{(1+v)(1-2v)} \begin{bmatrix} 1-v & v & v & 0 & 0 & 0 \\ v & 1-v & v & 0 & 0 & 0 \\ v & v & 1-v & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.5-v & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5-v & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5-v \end{bmatrix} \quad (3.12)$$

V_e = element volume

CHAPTER 4

RESULTS AND DISCUSSION

4.1 INTRODUCTION

This chapter will discuss on the results acquired through different type of analysis that had been performed in this chapter. The analysis was based on the impact stress on human synthetic bone. Purpose of performing the analysis is to determine the maximum stress the human synthetic bone able to withstand before fracture so a better material properties can be developed. The analysis will not cover the shape and size of human synthetic bone which can produce different result. However, the most important part by performing the analysis was to produce better product.

The impact stress analysis on human synthetic bone was performed using Finite Element Analysis Algor and MSC. Marc. The finite element model analysis is a computational technique used to obtain approximate solution of boundary value problems. The finite element analysis was also performed using different meshing, by considering the geometry sizes of the element used for analysis and making a comparison between manual calculation. The results collected will be displayed in graph form.

4.2 COMPUTER AIDED MODEL

The model geometry was develop in Solidwork which would be imported into MSC. Marc and Algor respectively for FEA later. There are three stages of development before the final design is decided. The finite element model has 12.5 percent dimension of the actual size. Figure 4.1 shows the initial geometry.

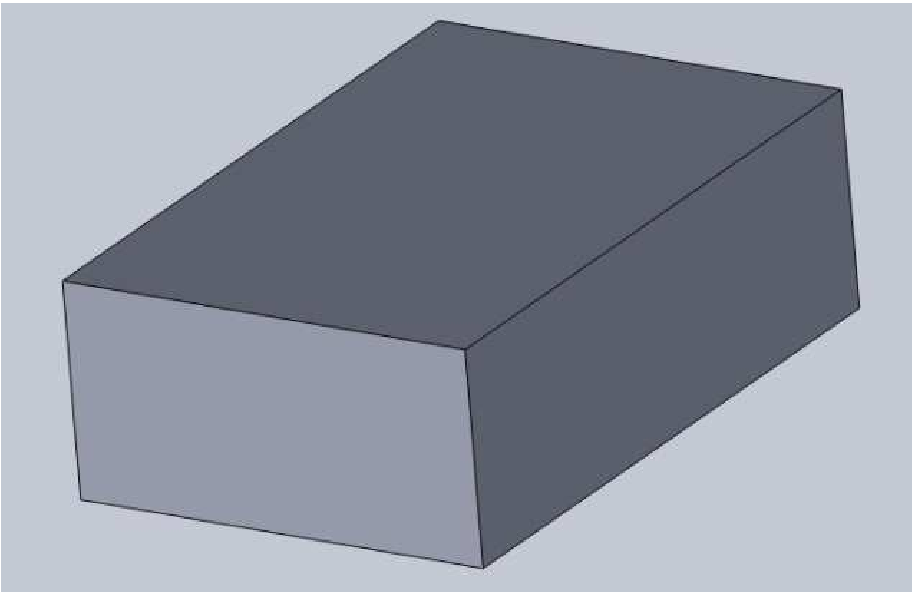

Figure 4.1: Initial geometry

A solid rectangular of 2mm x 4mm x 6mm. Pores of average size 200-300 μm was created on three orthogonal surfaces using spline curves. Internal pores were generated by random extrusion. Crucial part in construction of model geometry is creation of randomly-distributed porosity in the geometry and scaling down to meshable size. Since trial-and-error was the only option to create the model close to real one, it has taken a couple weeks to obtain a computable geometry.

4.2.1 Design 1

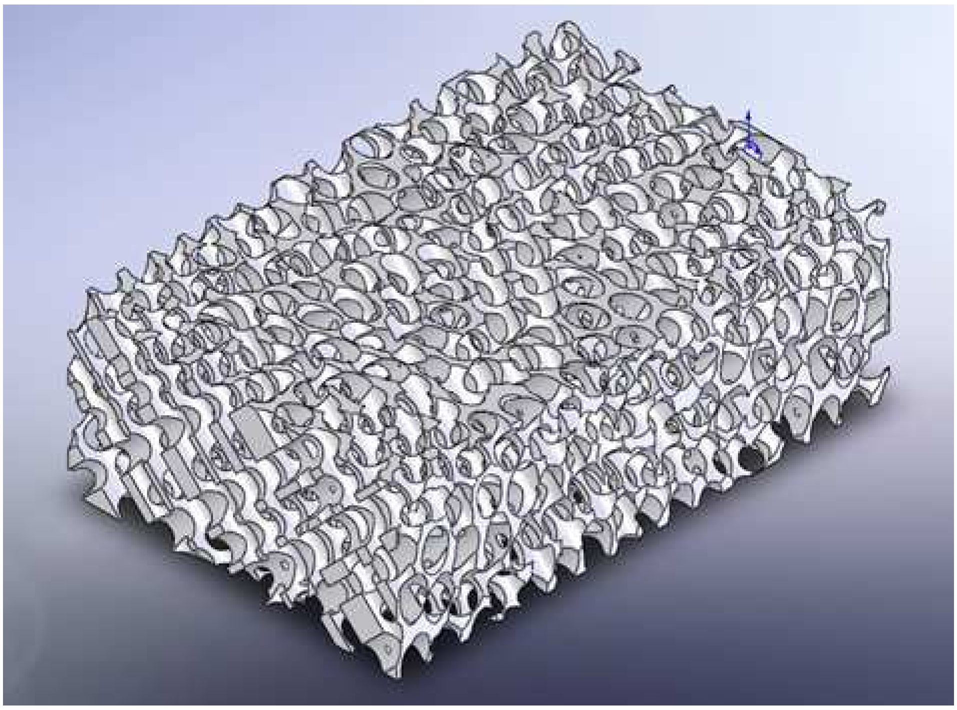

Figure 4.2: Rectangular geometry with porosity

Figure 4.2 shows design 1, a rectangular geometry having randomly-distributed pores. Several designs were created. Figure 4.3 and 4.4 are result of design 2 and 3 respectively after scaling down.

4.2.2 Design 2

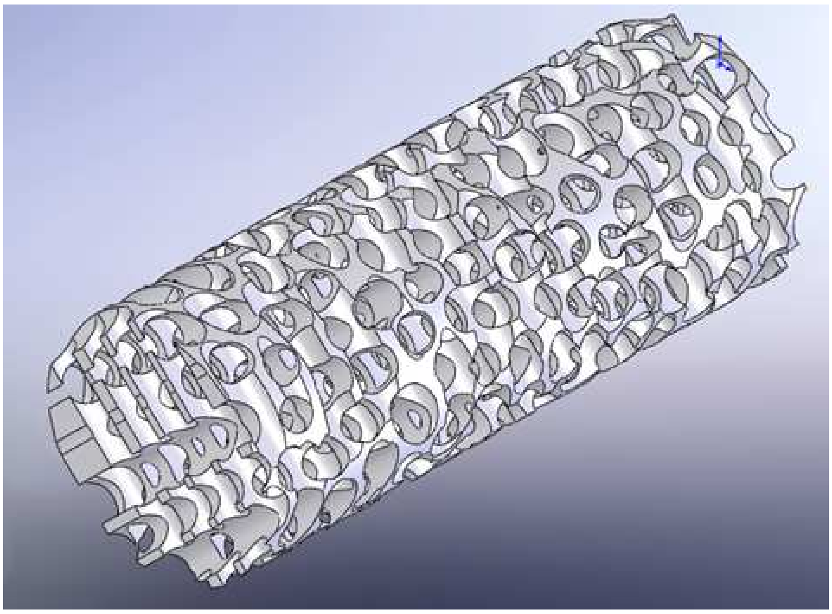

Figure 4.3: Design 2

As seen in Figure 4.3, design 2 is a cylinder with dimension of π x 1mm x 1mm x 5mm. Design 3 was similar to Design 2 since it has a smaller dimension for analysis purpose. Design 3 has a dimension of π x 0.5mm x 0.5mm x 2.5mm.

4.2.3 Design 3

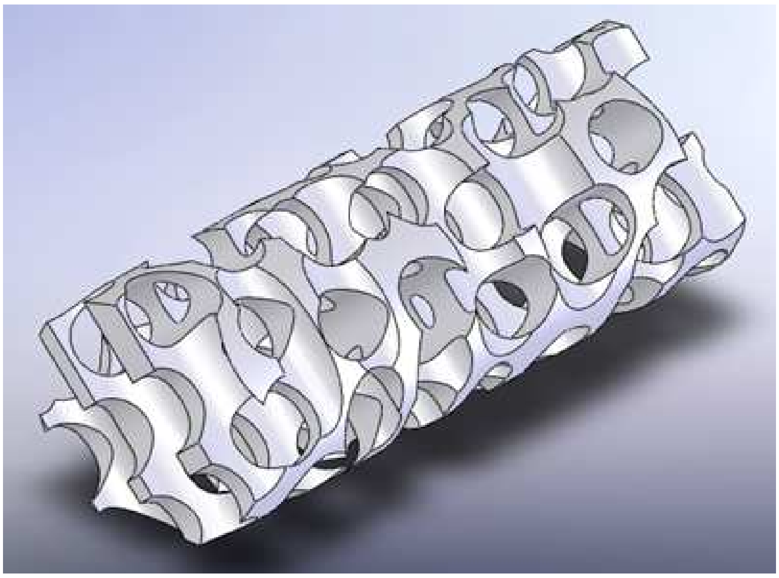

Figure 4.4: Design 3

4.3 PENDULUM GEOMETRY

Pendulum model geometry was designed in Solidwork with 12.5 percent of the real dimension to adjust to the scale down model of the bone model geometry. This model geometry was later used to simulate impact analysis to provide better resemblance to the real situation of impact test.

Figure 4.5: Pendulum real size

4.3.1 Design 1

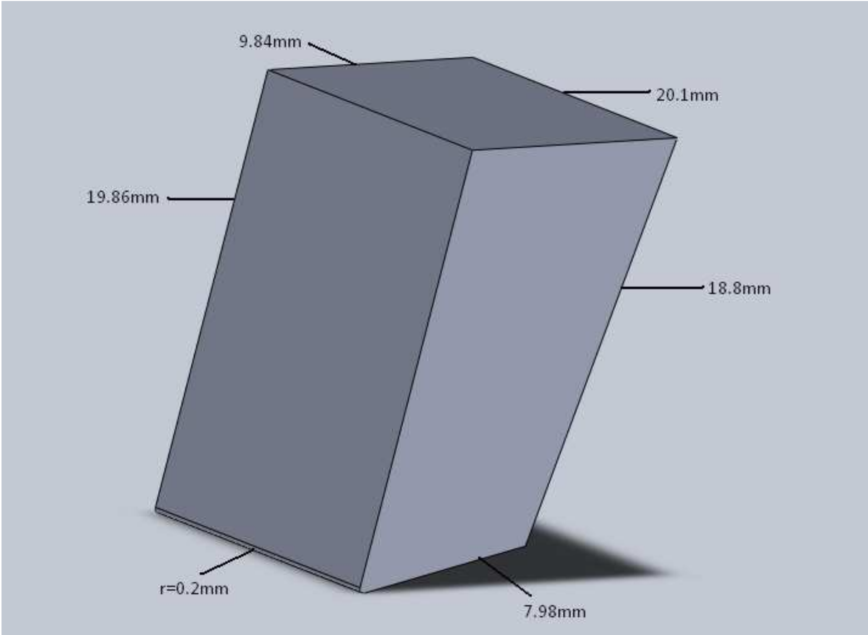

Figure 4.6: Design 1 for pendulum

Design 1 has the original dimension of the real pendulum head.

4.3.2 Design 2

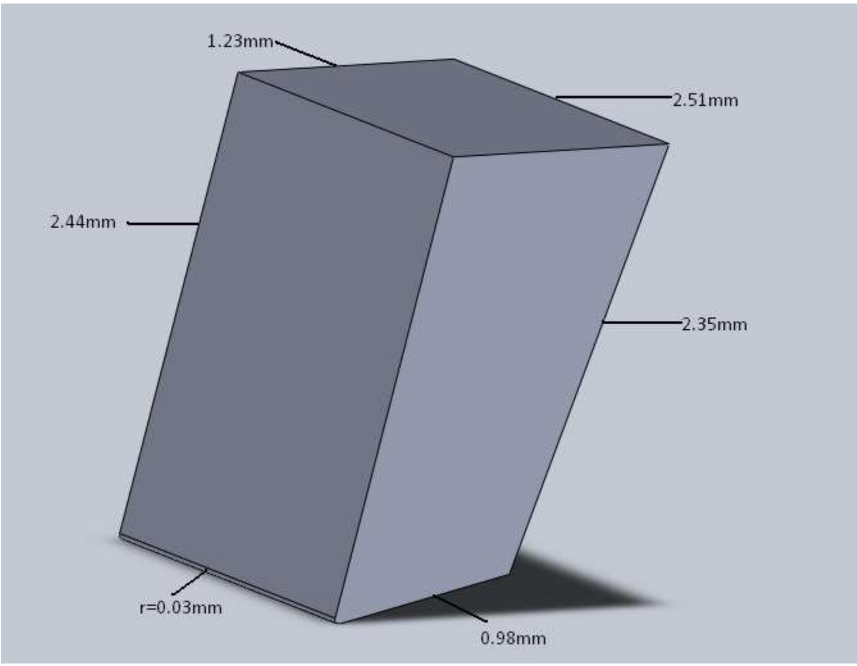

Figure 4.7: Design 2 for pendulum

Design 2 is the scale down model of the original dimension by 12.5 percent in order to be proportional with the bone model.

4.4 FINITE ELEMENT MODEL

The model geometry of the bone and pendulum is meshed with different meshing size respectively in MSC. Marc and Algor. Figure 4.8 shows the meshing of bone model geometry in Algor using tetrahedral elements while figure 4.9 shows the meshing pendulum model geometry in Algor using tetrahedral elements. Figure 4.10 shows finite element model geometry to analyze in Algor.

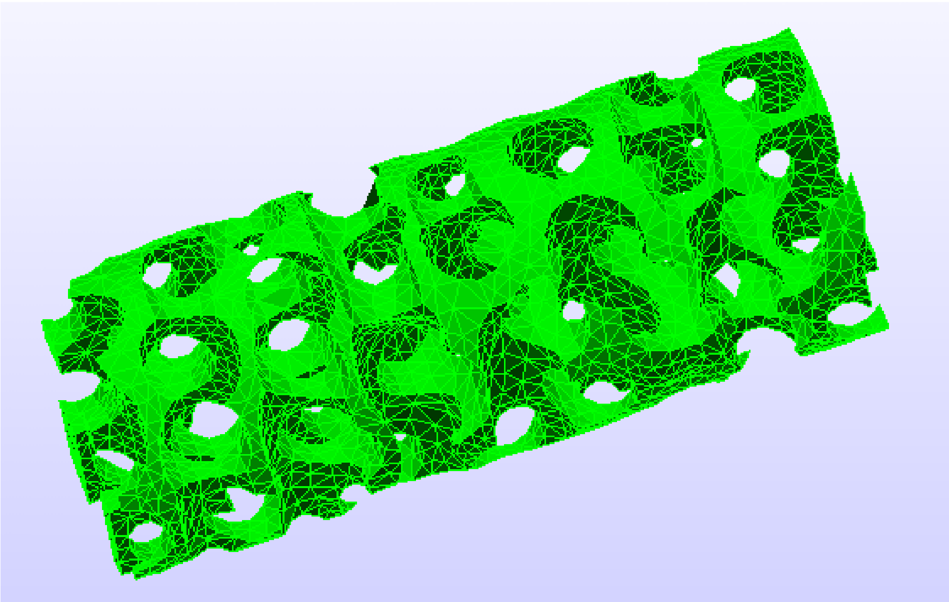

Figure 4.8: Bone meshing with 120% mesh density

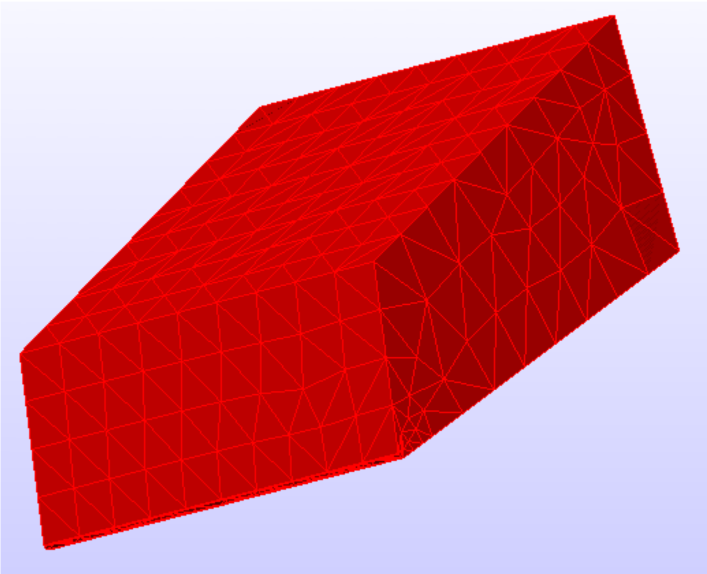

Figure 4.9: Pendulum with mesh density 100%

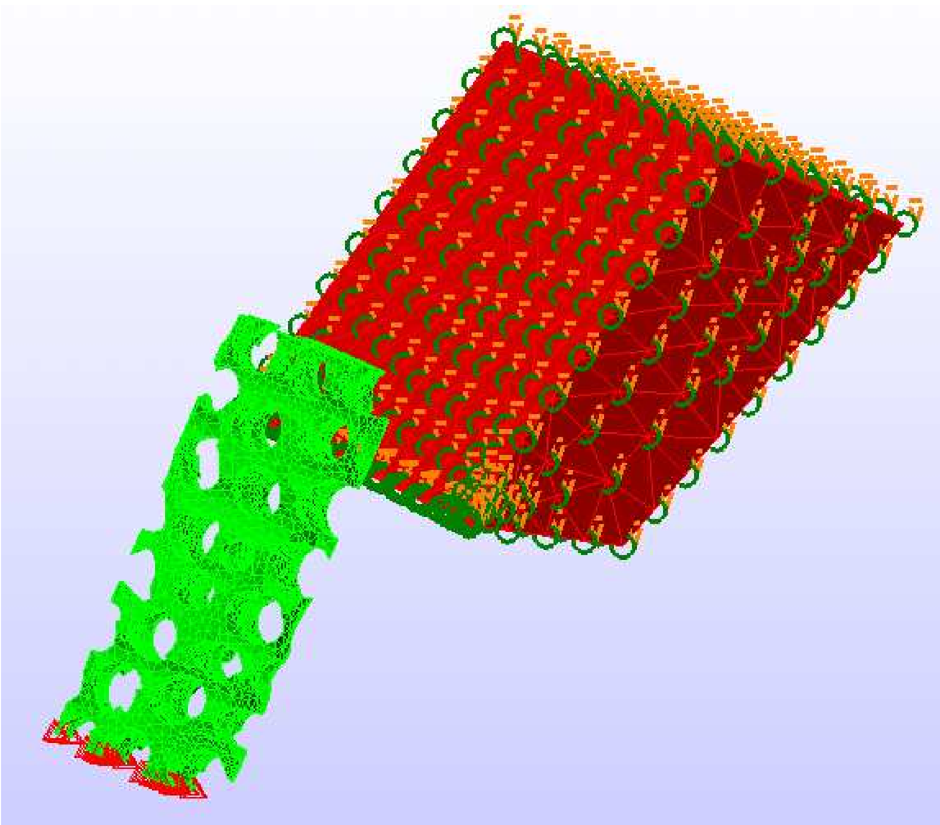

Figure 4.10: Finite element model geometry in Algor

Ib MSC. Patran, tetrahedral meshing was used to mesh the bone model geometry. Figure 4.11 shows the model geometry to analyze in MSC. Marc with 35968 elements.

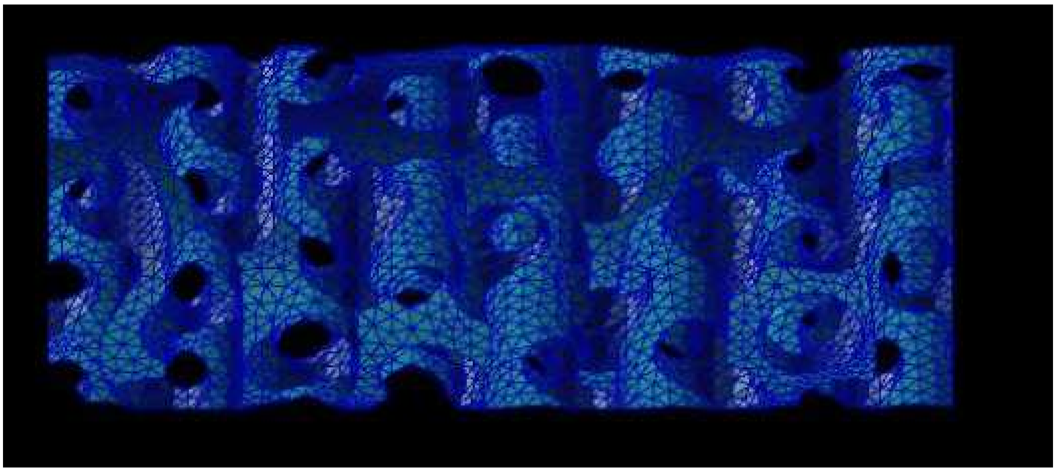

Figure 4.11: Finite element model geometry in MSC. Patran

4.5 BOUNDARY CONDITION

Boundary condition is applied on the model geometry such as displacement, velocity, and degree of freedom. For Algor, six degree of freedom is constrained at the bottom nodes of the bone. Figure 4.12 shows the boundary condition for the bone in Algor.

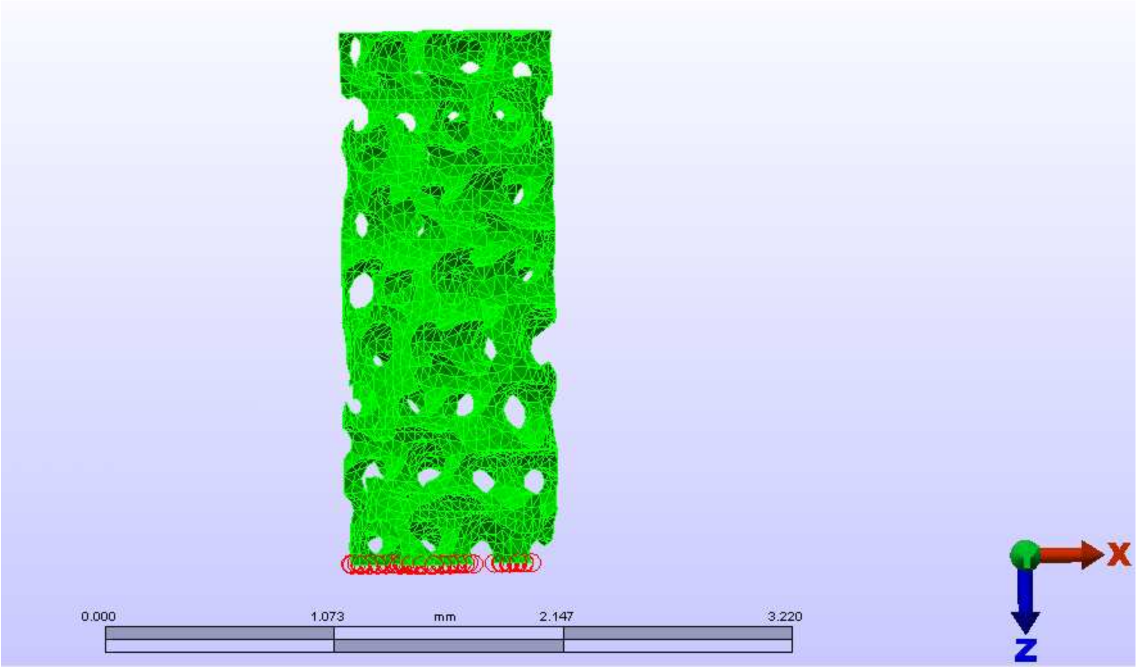

Figure 4.12: Boundary condition for bone.

Meanwhile for the pendulum model geometry, initial velocity of 3000m/s is applied to the part. Translation and rotation of all movement were constrained in the y and z direction. Figure 4.13 shows the boundary condition for pendulum and Figure 4.14 shows the complete finite element model in Algor.

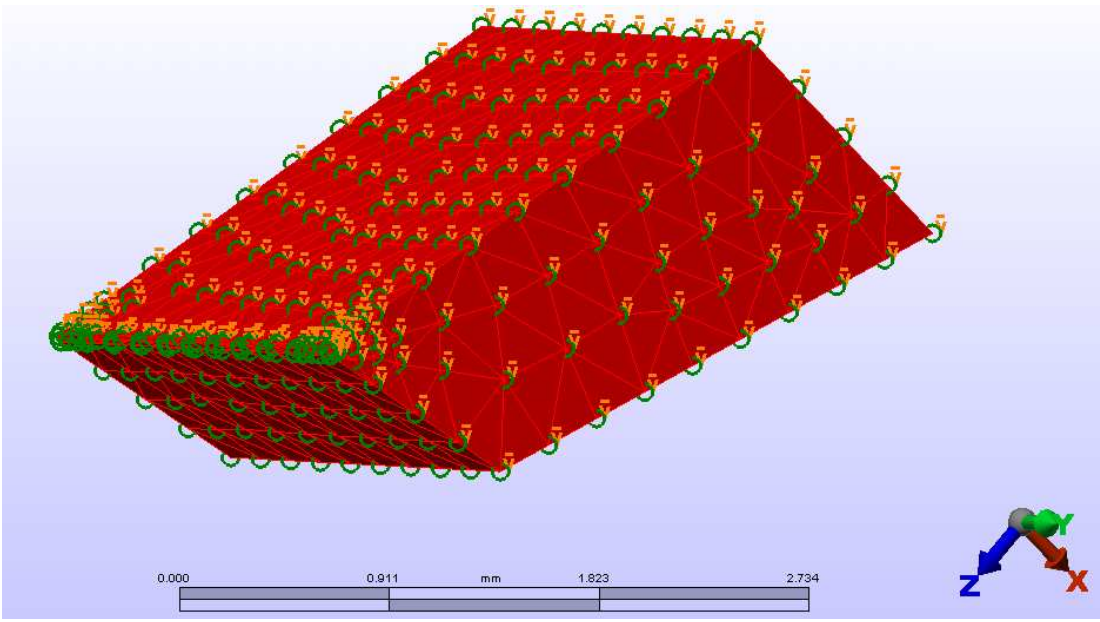

Figure 4.13: Boundary condition for pendulum

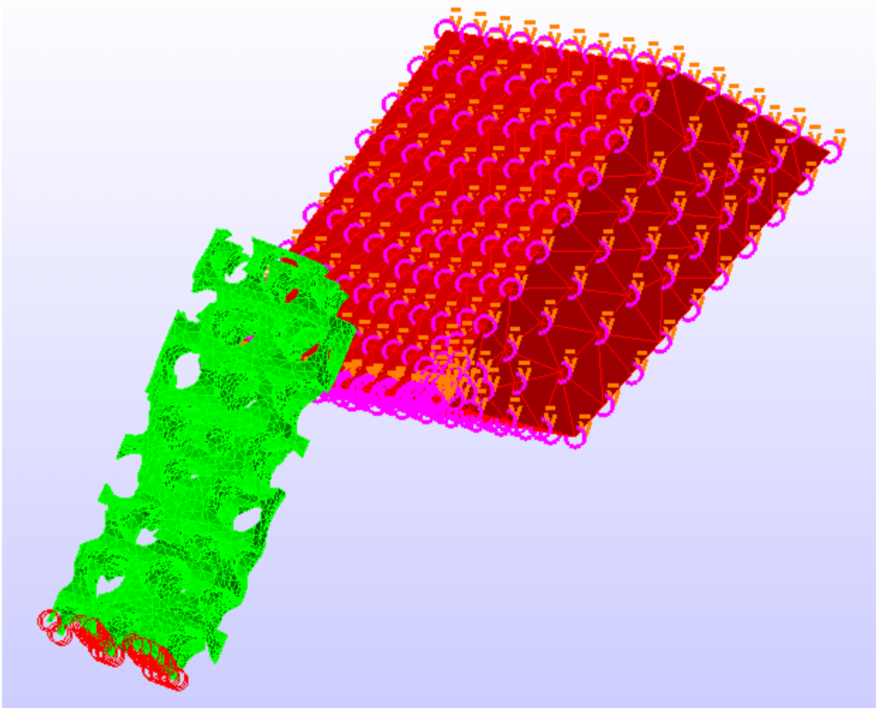

Figure 4.14: Complete model in Algor.

For MSC. Marc, all nodes at the one end of the surface is fixed and all nodes at another end of the surface are applied displacement of 0.3mm. Figure 4.15 shows the translation degree of freedom of all nodes at one end is constrained and Figure 4.16 shows displacement of 0.3mm at x-axis is applied on all nodes at the other end.

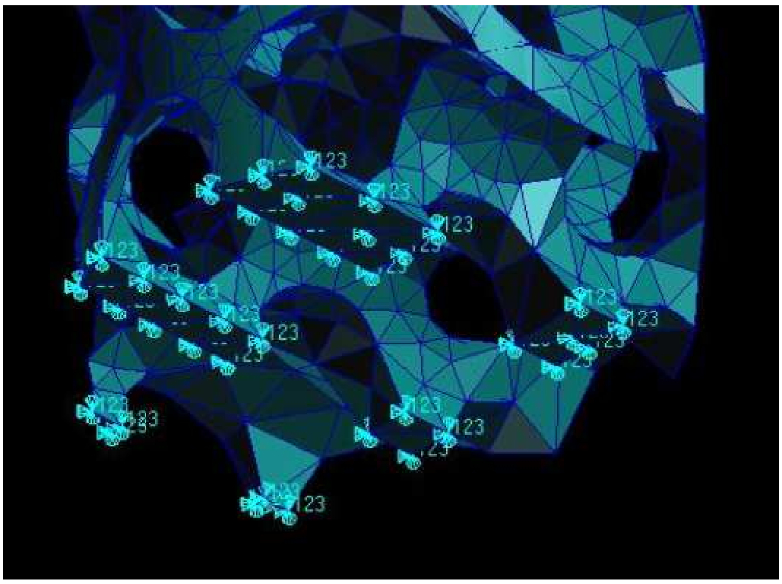

Figure 4.15: Constrained nodes.

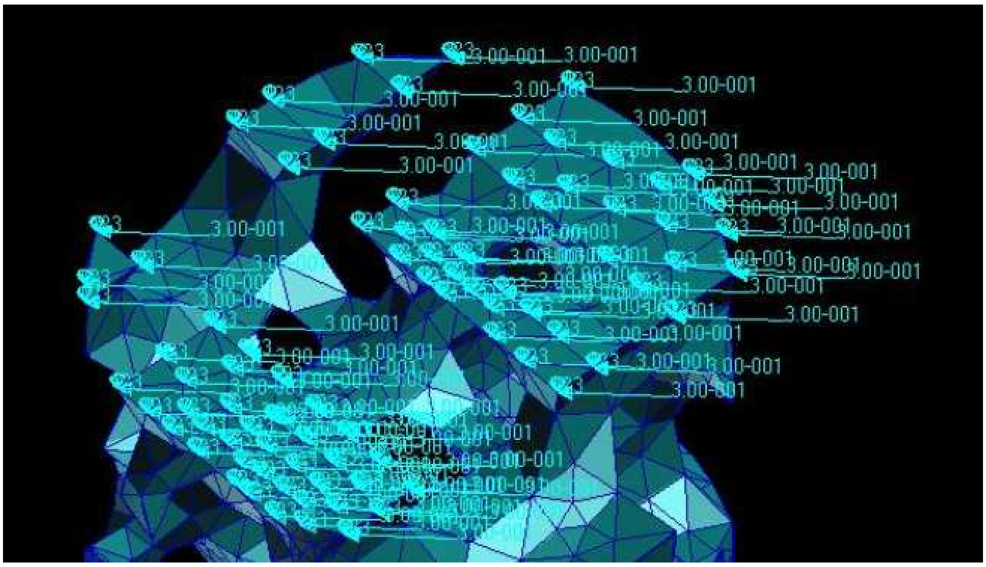

Figure 4.16: Displaced nodes.

4.6 ANALYSIS RESULT

There are numerous parameters involve in dynamic analysis using FEM. Some of those parameters were determined from a number of preliminary simulations. Some parameters such as damping factor, viscosity, minimum time steps were merely assume by default setting.

Principally, contour plots were extracted from each finite element analysis. The results will mainly focus on the von Mises Stress, von Mises Strain, Strain Energy, and Displacement Component. The following section discuss present and discuss analysis result from each analysis.

4.6.1 Results From MSC. Marc Analysis

The computation of displacement is verified in Figure 4.17 which shows the displacement contour. The maximum value was -0.3mm as shown at the top region of the model which is represented in white color. The value decreased as it approaches the lower side of the model. This result is consistent with the applied displacement as the applied nodal displacement is 0.3mm.

Figure 4.17: Displacement of X component

Figure 4.18 shows predicted von Mises stress contour. This result is reasonable in the sense that region of maximum stress is located nearly the applied impact load. In addition, the line of maximum stress is found to be slightly inclined to the x-axis.

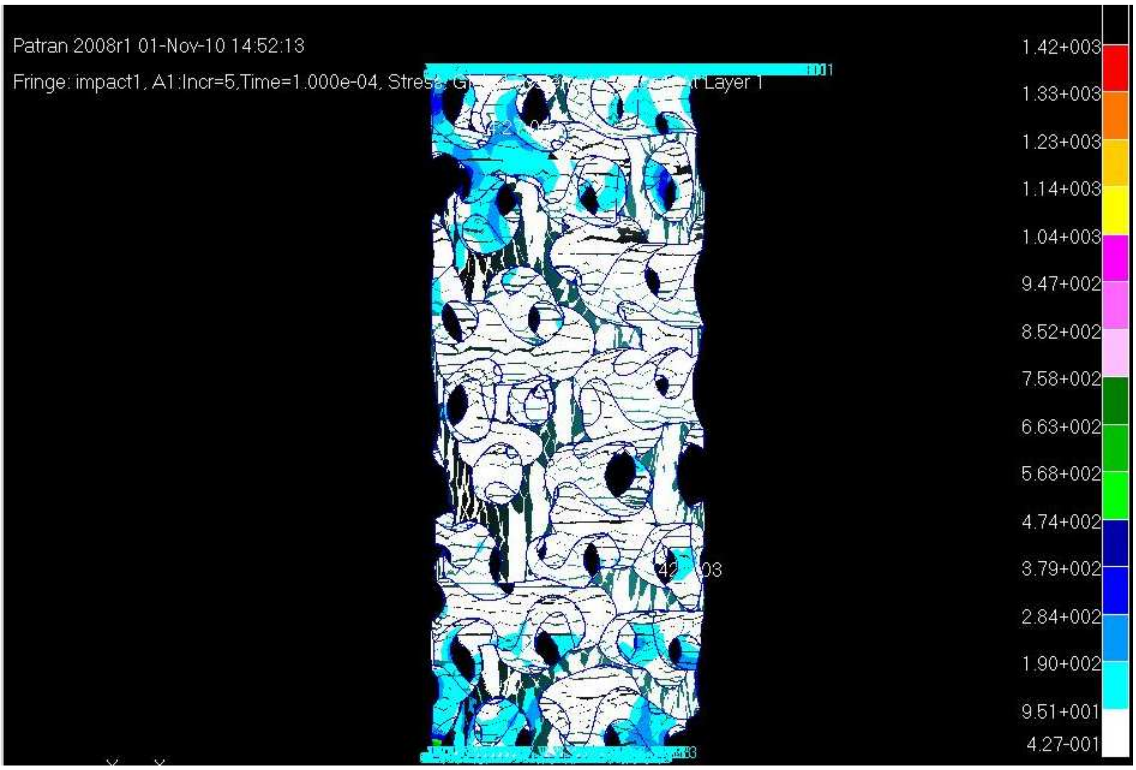

Figure 4.18: Von Mises stress

This is acceptable as there is randomly-distributed porosity in the geometry which deviates maximum stress from direction of applied load. Maximum stress is found to be 1420MPa.

Figure 4.19 shows predicted von Mises strain contour. Again, the line of maximum strain is found to be slightly inclined along the x-axis. Max strain is 0.277. The contour overall is similar with von Mises stress which proves that strain is linear with stress for this case.

Figure 4.19: Von Mises strain

Figure 4.20 shows the strain energy which is equivalent to energy absorbed by the bone during the impact. The highest value is 0.272J. This value is used to later validate the MSC. Patran finite element model.

Figure 4.20: Strain energy

4.6.2 Results From Algor Analysis

Figure 4.21 shows the computed displacement result. The maximum displacement was the pendulum travelled distance for the impact analysis.. This result is consistent with the applied displacement.

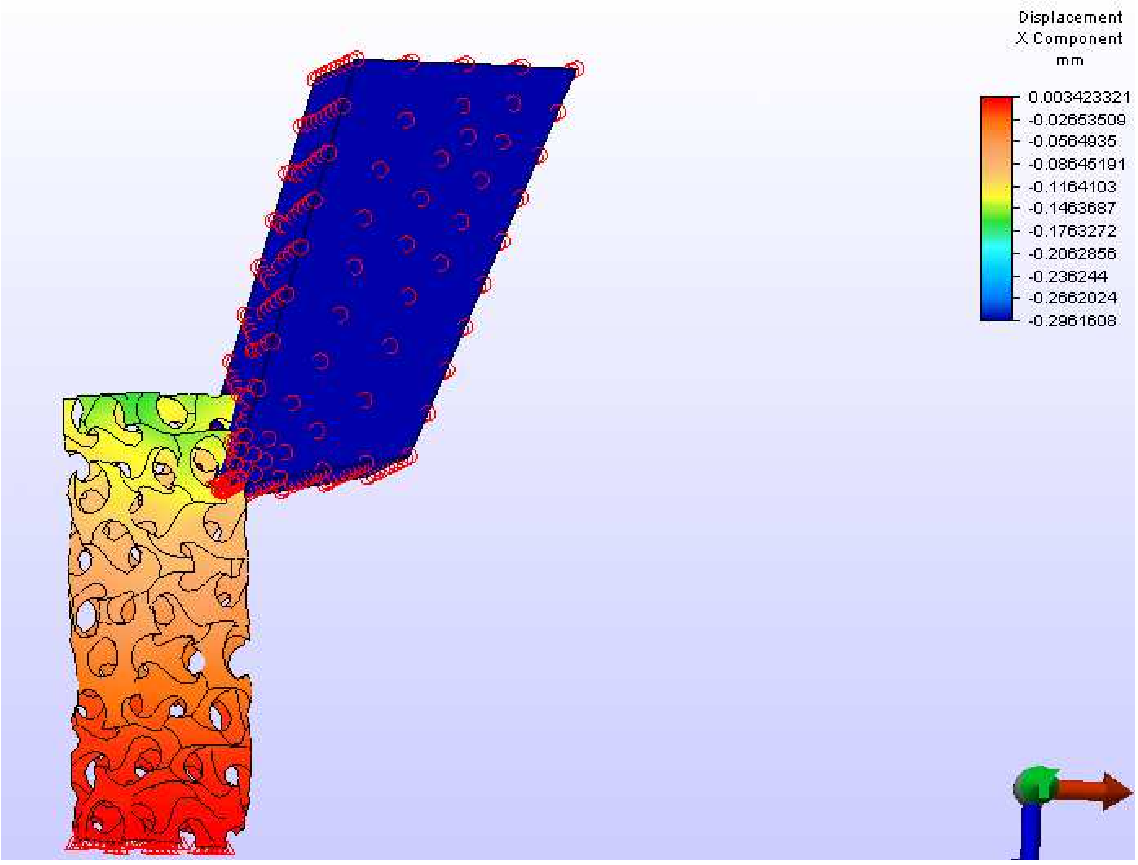

Figure 4.21: Displacement of X component.

Figure 4.22 shows predicted von Mises stress contour. The maximum stress contour is located at the surface of impact. This result is reasonable with the direction of applied impact. However, the stress values are lower than predicted for impact load.

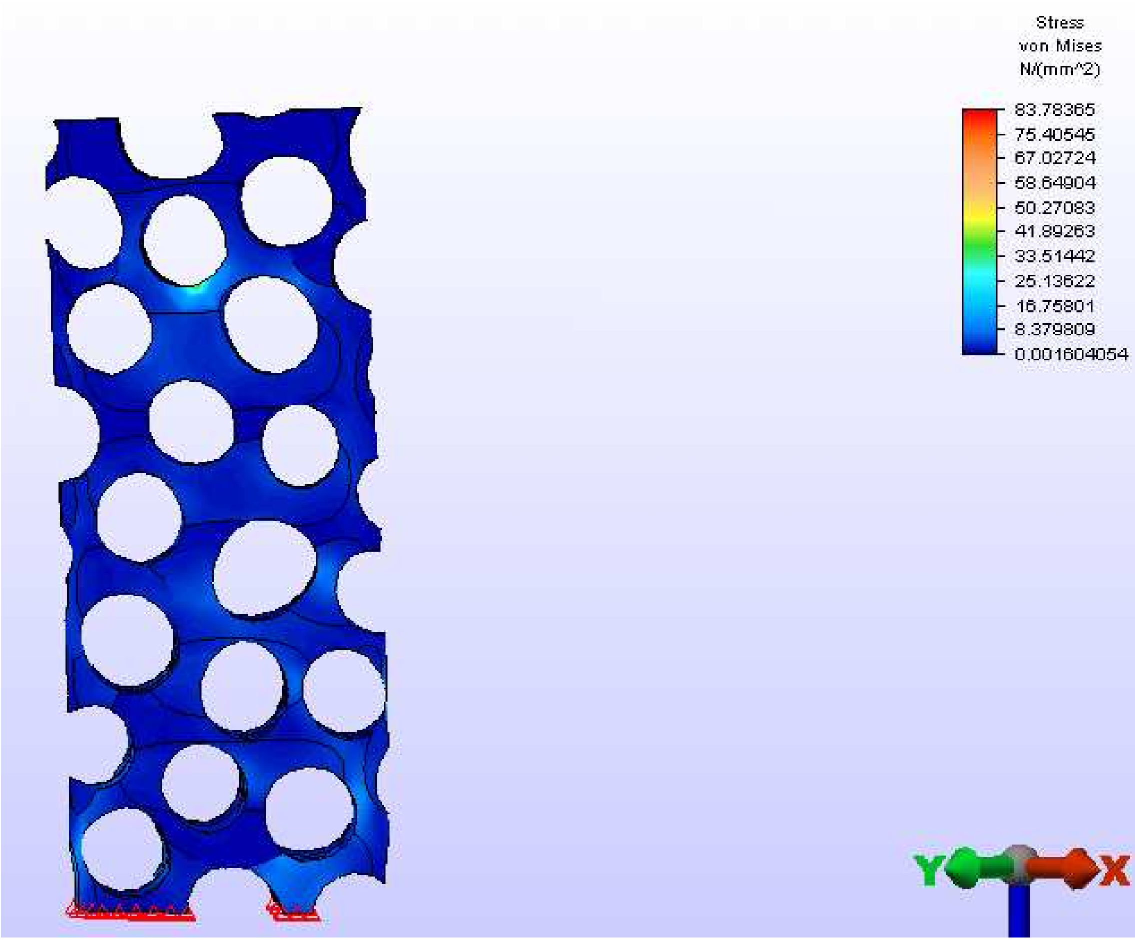

Figure 4.22: Von Mises stress

Figure 4.23 shows predicted von Mises strain contour. Similarly, the maximum strain contour is located nearby the applied impact. Also, the overall contour is similar with von Mises stress meaning stress and strain are linearly related. Although the numerical figure of stress and strain are far from those in MSC. Marc analysis, stress-strain relationship are agreeable in one another analysis. Such stress-strain response is consistent with the published report (Mechanical Behaviour of Materials 2th edition).

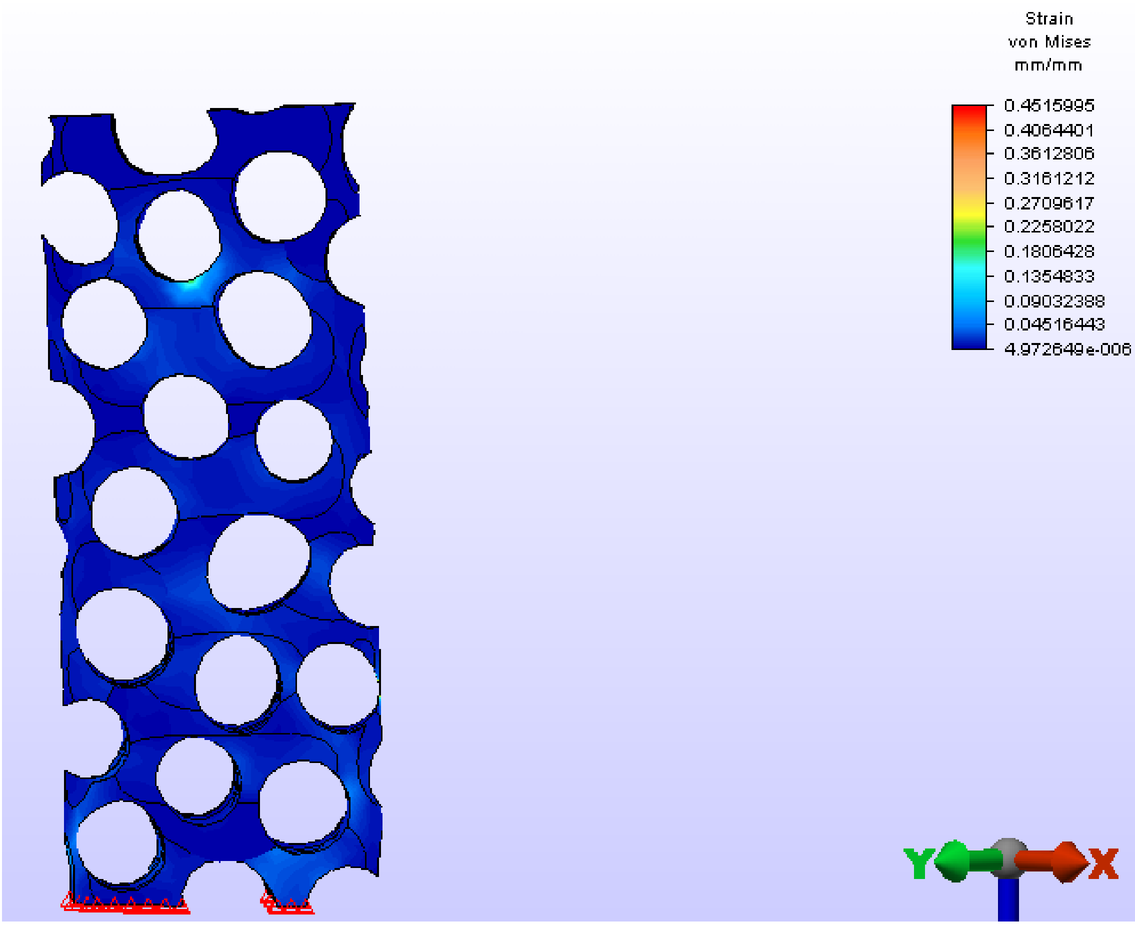

Figure 4.23: Von Mises strain.

4.7 MODEL VALIDATION

Result obtained from the analysis is compared to experimental value to validate the finite element model. Table 4.1 shows the comparison between predicted and experimented absorbed energy. Predicted energy was obtained in MSC. Marc analysis, whereas Algor analysis did not provide such output result. Consequently, the results were compared only for MSC. Marc. The impact model developed in Patran and analyzed by MSC. Marc is to be validated as error percentage which is less than 5%.

Table 4.1: Energy absorbed comparison

	Experimental	MSC. Marc
Impact energy absorbed (J)	0.28	0.272
Error (%)	-	2.86

On the other hand, the model analyze in Algor is verified by calculated stress value from absorbed energy. Table 4.2 shows the comparison of stress value. Stress calculation is provided in the appendix. Error in stress is acceptable for Algor model as the error percentage is less than 5%. Overall the model developed in MSC. Patran and analyzed by MSC. Marc is more agreeable to experimental results.

Table 4.2: Stress von Mises comparison

	Calculation	Algor
Stress (Mpa)	51.709	50.27
Error (%)	-	2.783

As for Algor model, some assumptions made in the material model is too approxiamate. This definitely affects the analysis results. The better way is to determine the model parameters experimentally, which is beyond the scope of the project. However, in terms of model geometry and finite element formulation, Algor model is more realistic.

From the project scope, stress-strain graph relationship is developed. The curve plotted can used to validate the result and shows material properties. Figure 4.24 shows the graph of stress-strain using 5 interval time. Graph plotted value is provided in the Appendix.

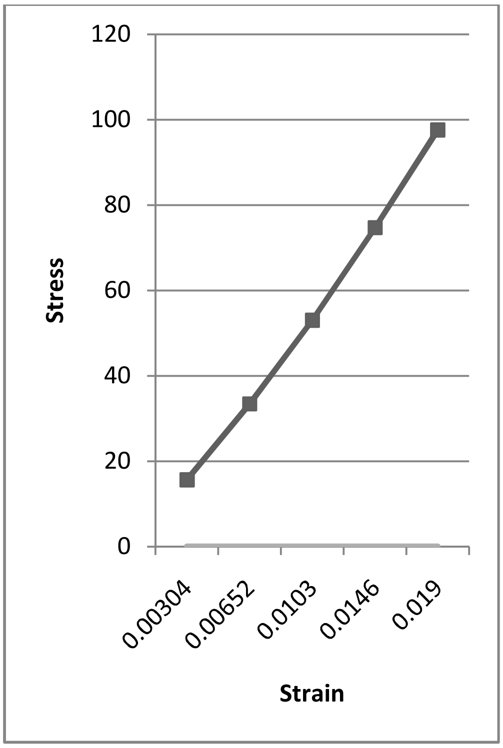

Figure 4.24: Stress-strain graph

From the graph plotted, stress and strain has linear relationship which is consistence with published result. Thus, the result obtained is validated.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 INTRODUCTION

In this chapter it will include with the overall. The results or outcomes of act or process will be concluded as the final judgment reached after consideration. Recommendations are also mentioned in this chapter for future improvement or development for this project.

5.2 CONCLUSION

The finite element model has been successfully developed to analyze the impact response of the human synthetic bone. From analysis results, the impact strength of synthetic bone can be predicted. The result obtained from both MSC. Marc and Algor analysis is validated with experimental value. Stress-strain response of bone specimen shows that stress is linearly related to strain with no apparent plastic strain. Of the finite element models developed and analyzed, Patran/Marc model provides reasonable results. This model has potential to use for further analysis where experimental analysis is prohibited.

5.3 RECOMMENDATIONS

There are many recommendations which can be done in order to improve the results. The recommendations which can be taken into consideration are listed below:

1. Advance learning on finite element software because there are still many parameters unexplored which could yield a more accurate result.

2. An improved model geometry which resembles the synthetic bone better.

3. Specific a more accurate contact simulation.