

RESPONSE OF HUMAN BONE SYNTHETIC
UNDER IMPACT LOAD USING
FINITE ELEMENT METHOD

MOHAMAD SHAHNAZ BIN SAMSUL BAHARI

BACHELOR OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled Response of human bone synthetic under impact load using finite element method is written by Mohamad Shahnaz Bin Samsul Bahari . 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 fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering.

Examiner

Signature

RESPONSE OF HUMAN BONE SYNTHETIC UNDER IMPACT LOAD USING
FINITE ELEMENT METHOD

MOHAMAD SHAHNAZ BIN SAMSUL BAHARI

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

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

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: Supervisor

Date: November 2009

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 concurrently submitted for award of other degree.

Signature:

Name: Mohamad Shahnaz Bin Samsul Bahari

ID Number: MA06050

Date: November 2009

COPYRIGHT

Copyright in text of this thesis rests with the author. Copies (by any process) either in full, or of extracts, may be made only in accordance with instructions given by the author and lodged in UMP. Details may be obtained from the librarian. This page must form part of any such copies made. Further copies (by any process) of copies made in accordance with such instructions may not be made without the permission (in writing) of the author.

The ownership of any intellectual property rights which may be described in this thesis is vested in UMP, subject to any prior agreement to the contrary, and may not be made available for use by third parties without the written permission of the University, which will prescribe the terms and conditions of any such agreement.

Further information on the conditions under which disclosures and exploitation may take place is available from the head of Mechanical Engineering.

I humbly dedicated this thesis to

my lovely father and late mother, Samsul Bahari Bin Abd Razal and Zarah Binti

Sulaiman

and my siblings

who always trust me, love me and had been a great source of support and motivation.

ACKNOWLEDGEMENTS

This thesis is the end of my long journey in obtaining my Bachelor Degree in Mechanical Engineering. I have been accompanied and supported by many people in this journey. It is a pleasant aspect that I have now the opportunity to express my gratitude for all of them.

First a very special thanks you to my supervisor, Dr. Daw Thet Thet Mon for her great patient and efforts to explain things clearly. She provided encouragements and lots of good ideas in order for me to complete the analysis and thesis writing period. I would have been lost without her guidance and support.

I would also indebted to the support of my family and friends. They accompanied me went through the toughest moment. A journey is easier when you travel together. Interdependence is certainly more valuable than independences.

ABSTRACT

Commercial bone synthetic can be considered as a composite material where testing can be carried out to find out its mechanical properties. Generally there is lack of information about the properties of bone synthetic although it has been extensively used in caring human bone defect and injuries. Particularly, its real strength to external load, particularly sudden load is unknown. This project focus on identify the response of human bone synthetic under impact loading using finite element method. The geometry of finite element model was constructed in SolidWorks mechanical CAD package. The micron size of pores was created spline-feature-extrude method to include porosity as close as possible to that of the actual bone. The computational model was analyzed using ALGOR. The appropriate model was determined through the mesh convergence analysis for better accuracy. The results were is good agreement in term of location of their concentration. The proposed finite element model can be used to analyze the response of the human bone under various loading.

ABSTRAK

Komersial tulang sintetik boleh dianggap sebagai material komposit di mana pengujian boleh dilakukan untuk mengetahui sifat mekanik nya. Secara umumnya terdapat kurangnya maklumat tentang sifat-sifat tulang sintetik meskipun telah banyak digunakan dalam merawat tulang manusia cacat dan luka-luka. Terutama, kekuatan sebenarnya beban luaran, terutama beban yang tidak diketahui. Projek ini tertumpu pada respon mengenalpasti tulang sintetik manusia di bawah pengaruh beban menggunakan *'finite element method'*. Geometri model dibuat dalam *'Solidwork CAD'* pakej. The mikron saiz porositi dibuat menggunakan kaedah *'spline-feature-extrude'* untuk menjadikan porositi sehampir mungkin dengan tulang yang sebenarnya. Model yang digunakan dianalisis dengan menggunakan *'ALGOR'*. Model yang sesuai ditentukan melalui analisis *'mesh convergence'* untuk mendapat akurasi yang lebih baik. Keputusan adalah baik dimana beban-beban adalah tertumpu sama seperti keputusan experiment. Model *'finite element'* boleh digunakan untuk menganalisis respon daripada tulang manusia dalam pelbagai beban.

TABLE OF CONTENTS

	Page
SUPERVISOR'S DECLARATION	ii
STUDENT'S DECLARATION	iii
COPYRIGHT	iv
ACKNOWLEDGEMENTS	vi
ABSTRACT	vii
ABSTRAK	viii
TABLE OF CONTENTS	ix
LIST OF TABLES	xii
LIST OF FIGURES	xiii
LIST OF ABBREVIATIONS	xiv
LIST OF NOMENCLATURE	xv
CHAPTER 1 INTRODUCTION	
1.1 Overview	1
1.2 Problem Statement	2
1.3 Objectives	2
1.4 Project Scopes	2
1.5 Organisation of this thesis	3
CHAPTER 2 LITERATURE REVIEW	
2.1 Introduction	4
2.2 Human Bone	4
2.2.1 Structure and Characteristic of Human Bone	5
2.3 Bone Synthetic	6
2.3.1 Bone Synthetic Materials	7
2.3.2 Autogenous Bone Graft	7
2.3.3 Allograft Bone Graft	7
2.4 Impact Loading	8
2.5 Finite Element Method	9

2.5.1	Nodes and Elements	10
2.5.2	Basic Step in Finite Element Analysis	10
2.5.3	Advantages of Finite Element Analysis	11
2.6	Test and Analysis on Human Bone Using FEM	11

CHAPTER 3 METHODOLOGY

3.1	Introduction	14
3.2	The Whole Project Flow	16
3.2.1	Literature review	16
3.2.2	Identifying the Objective, Problem and Project Scope	16
3.2.3	Methodology Detail	16
3.2.4	Development of Bone Geometry in Solidworks	16
3.2.5	Finite Element Analysis in ALGOR	17
3.2.6	Validation of Finite Element Analysis with Experiment	17
3.2.7	Report Writing	17

CHAPTER 4 RESULTS AND DISCUSSION

4.1	Introduction	18
4.2	Development of Bone Geometry in Solidworks	18
4.3	Finite Element Analysis	21
4.4	Mesh Convergence	29
4.5	Computed Result With 100% mesh size	29
4.6	Computed Result With 90% mesh size	31
4.7	Computed Result With 80% mesh size	33
4.8	Convergency Analysis	35
4.9	Bone Response	36
4.10	Model Validation	39

CHAPTER 5 CONCLUSION AND RECOMMENDATIONS

5.2	Conclusion	41
5.3	Recommendations	42

REFERENCES	43
APPENDICES	45
A1 Gantt Chart For FYP1	45
A2 Gantt Chart For FYP2	46

LIST OF TABLES

Table No.	Title	Page
4.1	Material Properties Used for Drucker-Prager	25
4.2	Pendulum energy and pendulum length	25
4.3	Result for the mesh convergence	35
4.4	Stress Von Mises Value for Node 5622	36
4.5	Strain Von Mises values for Node 5622	37

LIST OF FIGURES

Figure No.	Title	Page
2.1	Human Bone Structure	6
2.2	Micro Finite Element Human Bone Model	13
3.1	Methodology Flow Chart	15
4.1	Rectangular bone model	19
4.2	Cylindrical bone models	20
4.3	Cylindrical bone model with reduced size	20
4.4	Bone model with 100% mesh size	22
4.5	Bone model with 90% mesh size	22
4.6	Bone model with 80% mesh size	23
4.7	Nodal and Boundary condition applied to 100% mesh size model	26
4.8	Nodal and Boundary condition applied to 90% mesh size model	27
4.9	Nodal and Boundary condition applied to 80% mesh size model	28
4.10	Stress Von Mises contour for bone model with 100% mesh model	29
4.11	Stress tensor Y-Y contour for bone model with 100% mesh size	30
4.12	Stress tensor Y-Z contour for bone model with 100% mesh size	30
4.13	Stress Von Mises contour for bone model with 90% mesh size	31
4.14	Stress Tensor Y-Y contour for bone model with 90% mesh size	32
4.15	Stress Tensor Y-Z contour for bone model with 90% mesh size	32
4.16	Stress Von Mises contour for bone model with 80% mesh size	33
4.17	Stress Tensor Y-Z contour for bone model with 80% mesh size	34
4.18	Stress Tensor Y-Y contour for bone model with 80% mesh size	34
4.19	Graph of Stress versus Times of node 5622	38
4.20	Graph of Strain versus Times of node 6522	38
4.21	Graph of Stress-Strain	39
4.22	Load concentrated in FEA model	40
4.23	Load concentrated in impact experiment	40

LIST OF ABBREVIATIONS

CAD	Computer Aided Design
FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Model
FYP	Final Year Project
MES	Mechanical Event Simulation

LIST OF NOMENCLATURE

E	Modulus of Elasticity
σ	Stress
ε	Strain
L	Length
D	Diameter
α	Ducker-Prager Yield Function Alpha
β	Ducker-Prager Yield Function Beta
C	Damping Coefficient
δy	Yield Stress
F	Force
N	Newton

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

Human bone can be considered as a composite material where testing can be carried out to find out its mechanical properties. Generally there is lack of information about the properties of bone. This situation arises because of few studies conducted on the human bone as it is not easy to get the specimen. Nowadays, human bone can be replaced by bone synthetic. Bone synthetic is claimed as successor to the real bone. The synthetic bones also are made from composite material. The importance application of bone synthetic is to repair and healing at the site of a bone defect. It is successfully used in medical field because synthetic bone is no risk of immune response and no risk of disease transmission.

Few researchers have used Finite Element Analysis (FEA) to find out properties of natural bone. Finite Element Analysis (FEA) plays a bigger role in the analysis and testing of material virtually using FEA software such as ALGOR, COSMOS and LS-DYNA. The benefits of using FEA are modeling of complex boundary condition and material models and realistic simulation of variety of problems. Besides that, the advantage of using FEA is that it is economical by reducing the number of experiment FEA allows more intricate analysis of material besides making it easy to conduct different tests to a material [Hutton, David V.,2004]. In this project FEA is used to find out the response of human bone synthetic under impact load.

1.2 PROBLEM STATEMENT

Commercial bone synthetic has been extensively used in caring human bone defect and injuries. However, its real strength to external load, particularly sudden load is unknown. Extensive experimental work is too expensive.

1.3 OBJECTIVE

The objectives of this project are:

- i. To develop a computational model of bone synthetic using Finite Element Method (FEM).
- ii. To simulate the response of bone synthetic under impact load.
- iii. To establish response curve for bone synthetic subjected to impact load.

1.4 PROJECT SCOPE

The scope of the project is:

- i. Model geometry will be created in Solidwork
- ii. Computational model will be developed using ALGOR.
- iii. Computational model will be validated with experiment result done by others.
- iv. Response of bone synthetic to impact load will be analyzed.
- v. Response curve will be developed from simulated results.

1.5 ORGANISATION OF THIS THESIS

This thesis consists of five chapters. Chapter one is highlights background of the project, problem statement, objectives and project scope. Chapter two is literature reviews which study about researchers has done before this and make it as references and guide line for this project. Chapter three is for the methodology which is the explanation for the flow and step of this project. Chapter four is for the result and discussion. In this chapter is discussing the result for the analysis and validated the result. The last chapter is chapter five which is conclusion and recommendation for this project.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

As a part of the project, the analysis of literature is important as to have a further understanding the project. The materials that used for the literature review are from the internet web pages, journals, and from the books. The review is to find out the relevance of the project and it must have a significant relation with the project.

2.2 HUMAN BONE

Bone is a hybrid of organic and inorganic composite containing protein and mineral with superior hardness, strength and fracture toughness [Huajian Gao, Baohua Ji, 2003]. It is a rigid organ which is a part of human endoskeleton, it functions to enable movement, provide support and also to protect vital organs as it has a very hard feature. Besides that, human bones also function to store minerals, produces red and white blood cells [Enderle, John D., Bronzino, Joseph D., and Blanchard, Susan M., 2005]. Bones has many shapes and has a very complex structure both internal and external. Bone tissue or osseous tissue makes up as the element of bone structure which gives its rigidity and honeycomb internal structure. [Wikipedia/Bone].

2.2.1 Structure and Characteristic of Human Bone

The bone is made up of calcium phosphate as its main structural material. Human bone is generally classified into two types that are: cortical (compact) bone and Trabecular or Cancellous bone [www.engin.umich.edu/class/bme456/bonestructure]. It is classified based on its porosity level and microstructure. Generally the compact bone or cortical bone has a very denser structure and its porosity level is ranging from 5% to

10%. From overall human body's bone mass, cortical makes about Cortical bone is very compact and has a very high strength. Thus it forms the outer layer around the Trabecular bone in shaft of long bones. Trabecular or cortical bone is much porous with porosity level in between 50% to 90%.It accounts for the remaining 20% of the total bone mass and has almost ten times of the surface area of compact bone. [Wikipedia/bone].

The cortical bone as other biological tissues has a hierarchical structure that is bone contains many different structures. On general it has been classified into three main type based on its microstructure organization.

Bone has up to seven hierarchical level of organization from the microstructure of the mineral crystals to the microstructure of cancellous and cortical bone [Huajian Gao, Baohua Ji, 2003].This mean the cortical bone has different structure layers similar to a Russian dolls. [Enderle, John D.,Bronzino, Joseph D., and Blanchard , Susan M., 2005].

The average trabecular Young's modulus measured ultrasonically and mechanically was 14.8 GPa (S.D. 1.4) and 10.4 (S.D. 3.5) and the average Young's modulus of microspecimens of cortical bone measured ultrasonically and mechanically was 20.7 GPa (S.D. 1.9) and 18.6 GPa (S.D. 3.5). With either testing technique the mean trabecular Young's modulus was found to be significantly less than that of cortical bone ($p < 0.0001$). [[Rho JY](#), [Ashman RB](#), [Turner CH](#), J biomech, 1993]



Figure 2.1: Human Bone Structure

2.3 BONE SYNTHETIC

Bone synthetic is a piece of bone used to take the place of a removed bone or bony defect. Bone grafting is a surgical procedure that replaces missing [bone](#) with material from the patient's own body, an artificial, synthetic, or natural substitute. Bone grafting is used to repair [bone fractures](#) that are extremely complex, pose a significant health risk to the patient, or fail to heal properly. [Wikipedia/Bone_grafting]

Every method of bone grafting has its own advantages and disadvantages (pain, scarring, infection, and cost), but there are many viable options and patients are well-served to be educated about their choices regarding this aspect of a spinal fusion, and proactively work together with their surgeon to select the option that is right for them [Ebraheim NA, Elgafy H, Xu R, 2001]

2.3.1 Bone Synthetic Materials

[Artificial bone](#) can be created from ceramics such as calcium phosphates ([hydroxyapatite](#) and [tricalcium phosphate](#)), [Bioglass](#) and [calcium sulphate](#); all of which are biologically active to different degrees depending on solubility in the physiological environment[Wikipedia/Bone_grafting]. These materials can be doped with [growth factors](#), ions such as [strontium](#) or mixed with [bone marrow](#) aspirate to increase biological activity. Some authors believe this method is inferior to autogenous bone grafting [Jeffery C, Wang, MD, Bone Graft: New Development, 2009].

Artificial synthetic bones are made of HTR polymer, which fosters the growth of new bone. Synthetic bone grafts are implemented in several sectors: if bone and periodontium defects are to be corrected, when the maintenance of bone tissue needs to be fostered, and in situations where the dental lamina needs to be raised [www.alfieridentista.it/eng/synthetic-bones-grafts.html].

2.3.2 Autogenous Bone Graft

The gold standard of bone grafting is taking the patient's own bone. This is called autogenous bone graft. This means that at the time of surgery, the surgeon makes a separate incision and takes a small piece of bone from an area of the body where it is not needed. Typically, autogenous bone grafts are taken from the pelvis or iliac crest. Autogenous bone grafting has excellent fusion rates and has become the standard by which all other biologics are measured. Many surgeons prefer autogenous bone grafts because there is no risk of the body rejecting the graft since it came from the patient's own body [Jeffery C, Wang, MD, 2009].

2.3.3 Allograft Bone Graft

In an effort to minimize the problems associated with taking the patient's own bone, a number of other fusion techniques have been developed that use biological products as bone graft extenders or as bone graft replacements. One common source of bone graft replacement or extender is the use of allograft bone. An allograft bone graft is bone harvested from cadavers or deceased individuals who have donated their bone for use in the treatment of living patients. This is commonly used in many forms for

spinal fusions ranging from cervical interbody fusions to lumbar interbody fusions and can provide excellent structural support [Jeffery C, Wang, MD, 2009].

2.4 IMPACT LOADING

Impact testing is testing an object's ability to resist high-rate loading. An impact test is a test for determining the energy absorbed in fracturing a test piece at high velocity.

Most ductile materials have strength properties which are a function of the loading speed. The more rapid loading, give the higher tensile and ultimate strengths of the materials. Two standard tests of the impact loading are the Charpy and Izod. This two test can measure the impact energy which is the energy required to fracture a test piece under an impact load. It also called the notch toughness.

The detailed assessment of the strength of machine elements under impact loading regimes involves use of advanced techniques including Finite Element Analysis. Impact loads result in shock waves propogating through the elements with possible serious consequences. It is possible to complete relatively simply stress evaluation for suddenly applied and impact loads by using the principle of conservation of energy and conditional that the materials considered are operating within their elastic regions. [www.roymech.co.uk/Useful_Tables/Fatigue/Mechanics_Impact.html].

Most real world impacts are biaxial rather than unidirectional. Further complication is offered by the choice of failure modes: ductile or brittle. Brittle materials take little energy to start a crack, little more to propagate it to a shattering climax. Other materials possess ductility to varying degrees. Highly ductile materials fail by puncture in drop weight testing and require a high energy load to initiate and propagate the crack.

Many materials are capable of either ductile or brittle failure, depending upon the type of test and rate and temperature conditions. They possess a ductile/brittle transition that actually shifts according to these variables. [web@instron.com]

2.5 FINITE ELEMENT METHOD (FEM)

The finite element method or FEM also referred to as finite element analysis is a numerical technique for finding approximate solutions of partial differential equations as well as of integral equations. Or simply, Finite element analysis is a computerized method for predicting how a real world object will react to forces, heat, vibration, and so on, in terms of where it will fail. It is called analysis, but in the product design cycle it is used to predict what is going to happen when the product is used. [Wikipedia/ FEM].

The finite element method works by breaking a real object down into a large number of elements, such as little cubes. The behavior of each little element, which is regular in shape, is readily predicted by set mathematical equations. The computer then adds up all of the individual behaviors to predict the behavior of the actual object. The finite in finite element analysis comes from the idea that there are a finite number of elements in a finite element model. Finite element method is employed to predict the behavior of things with respect to virtually all physical phenomena such as Mechanical stress, Mechanical vibration, Heat transfer, Fluid, Various electrical and magnetic phenomena and Acoustics. [Algor Help File, Algor corp].

The finite element method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the domain. Boundary values problems are also sometimes called field problems. The field variables are the independent variables of interest governed by the differential equation. The boundary condition is the specified values of the field variables or related variables such as derivatives on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only few. [Hutton, David V., 2004]

2.5.1 Nodes and Elements

A node is a coordinate location in space where the degrees of freedom are defined. The degree of freedom represents the possible movement of this point due to the loading of the structure. The DOFs also represent which forces and moments are transferred from one element to the next. The results of a finite element analysis - deflections and stresses, are usually given at the nodes. In the real world, a point can move in 6 different directions, translation in X, Y, and Z, and rotation about X, Y, and Z. In FEA, a node may be limited in the calculated motions for a variety of reasons. For example, there is no need to calculate the out of plane translation on a 2-D element; it would not be a 2-D element if its nodes were allowed to move out of the plane. [Hutton, David V., 2004].

An element is the basic building block of finite element analysis. There are several basic types of elements. An element is a mathematical relation that defines how the degrees of freedom of a node relate to the next. These elements can be lines- trusses or beams, areas- 2D or 3D plates and membranes or solids- bricks or tetrahedrals. It also relates how the deflections create stresses. [Widas, Peter., 1997]

2.5.2 Basic Step in Finite Element Analysis

The basic steps in a finite element analysis are divided into 3 major steps that are setting up the model, analyzing the model and results evaluation. Each step has its own division where for the setting up the model at first the model must be meshed, and then the units are defined. After that the model analysis parameter and the element type parameter are defined. Lastly for this step the load and the boundary conditions are applied. Move on to the second step where the assembly of the stiffness matrix is done followed by solving the linear algebraic equations. Then the results are calculated. For the third step results evaluation is done by reviewing the results and generate a report for the analysis.

2.5.3 Advantages of Finite Element Analysis

The development in the finite element analysis is really an advantage in engineering especially, in designing and manufacturing products. This is because FEM makes it easy to conduct test on products and materials virtually before even manufacturing the products. This allows the analysis can be done and the fault on the design or the material can be identified easily. Thus it is economical and more intricate analysis can be done.

2.6 TEST AND ANALYSIS ON HUMAN BONE USING FEM

There are several studies conducted for human bone properties using finite element and also conventional testing. The bones of a person with osteoporosis have become thin and fragile and are more likely to fracture. Predicting whether a human vertebra will fracture is a prodigious challenge due to the fact that numerous factors contribute to bone strength. Among these factors is the intrinsic micro damage in bone called micro cracks [Follet H, 2006]. Thus there is increasing evidence, however, bone responds in particular to a dynamic load. It has been shown that the application of high-frequency, very low magnitude strains to a bone can prevent bone loss due to osteoporosis and can even result in increased bone strength in bones that are already osteoporotic. In order to better understand this phenomenon, it is necessary to determine the strain as sensed by the bone cells due to this loading. This would be possible with the help of Finite Element Analysis. [Arbenz, P., 2006].

With the advancement of technology, powerful computers and simulation techniques are becoming popular for investigating the mechanical properties of bone. Using micro structural finite element (μ FE) models generated directly from computer reconstructions of trabecular and perform a 'virtual experiment', that is to simulate a mechanical test in great detail and with high precision.

For the clinician, predicting fracture risk for individual patients essentially amounts to the quantitative analysis of bone density. However, several studies have

shown that bone strength, an indicator for bone fracture risk, is only moderately predicted by bone density. This is not surprising: bones are not solid structures, but are made up of an outer shell of compact bone enclosing a core of spongy, trabecular bone. This trabecular bone, which is located at the end of long bones, has a porous structure that contributes significantly to the load-bearing capacity of the human skeleton. The investigation of the mechanical properties of trabecular bone presents a major challenge. This is due to its high porosity and complex architecture, both of which vary substantially between anatomic sites and across individuals. A promising technique that takes bone micro architecture into account is micro structural finite element- microFE analysis. A very large number of finite elements are needed to accurately represent a human bone with its intricate micro architecture; hence, the resulting microFE models possess a very large number of degrees of freedom. [Arbenz, P., 2006]

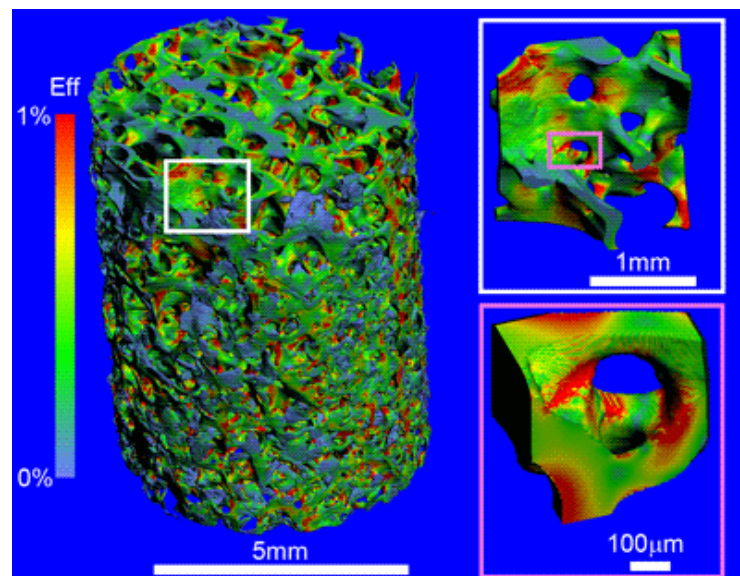


Figure 2.2: Micro Finite Element Human Bone Model

The figure 2.2 shows model of about 1.6 billion degrees of freedom was solved on an IBM BG/L. It is the largest model we solved so far. The visualization was done by the Data Analysis and Visualization Unit at CSCS, the Swiss Supercomputing

Centre. A further important issue is the visualization of the results. This work is done interactively and hence takes much more wall clock time than the simulation itself. The large data volume again requires parallel processing.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

The core of this project is to find out the response of human bone synthetic under impact load using Finite Element Method (FEM). Before going specific on this title, the process of study needs to know more detail about the knowledge of the human bone synthetic. Through the research and also journal that publish by the previous expertise, the processes to achieve the objective and solve the problem statement were helped. Find out the literature review also give the idea, how to develop of the model and learn advance in analysis of finite element model. From methodology flow chart review, the task will go step by step to find out the objectives of the project. For this project, there is three main parts to analysis the human bone synthetic. First is development of model geometry using Solidworks and then the model will be analyzing using ALGOR (FEMPRO). The last part of this project is validated the model that be analysis with the impact experiment which is done by other. Figure 3.1 show the flow chart of this project.

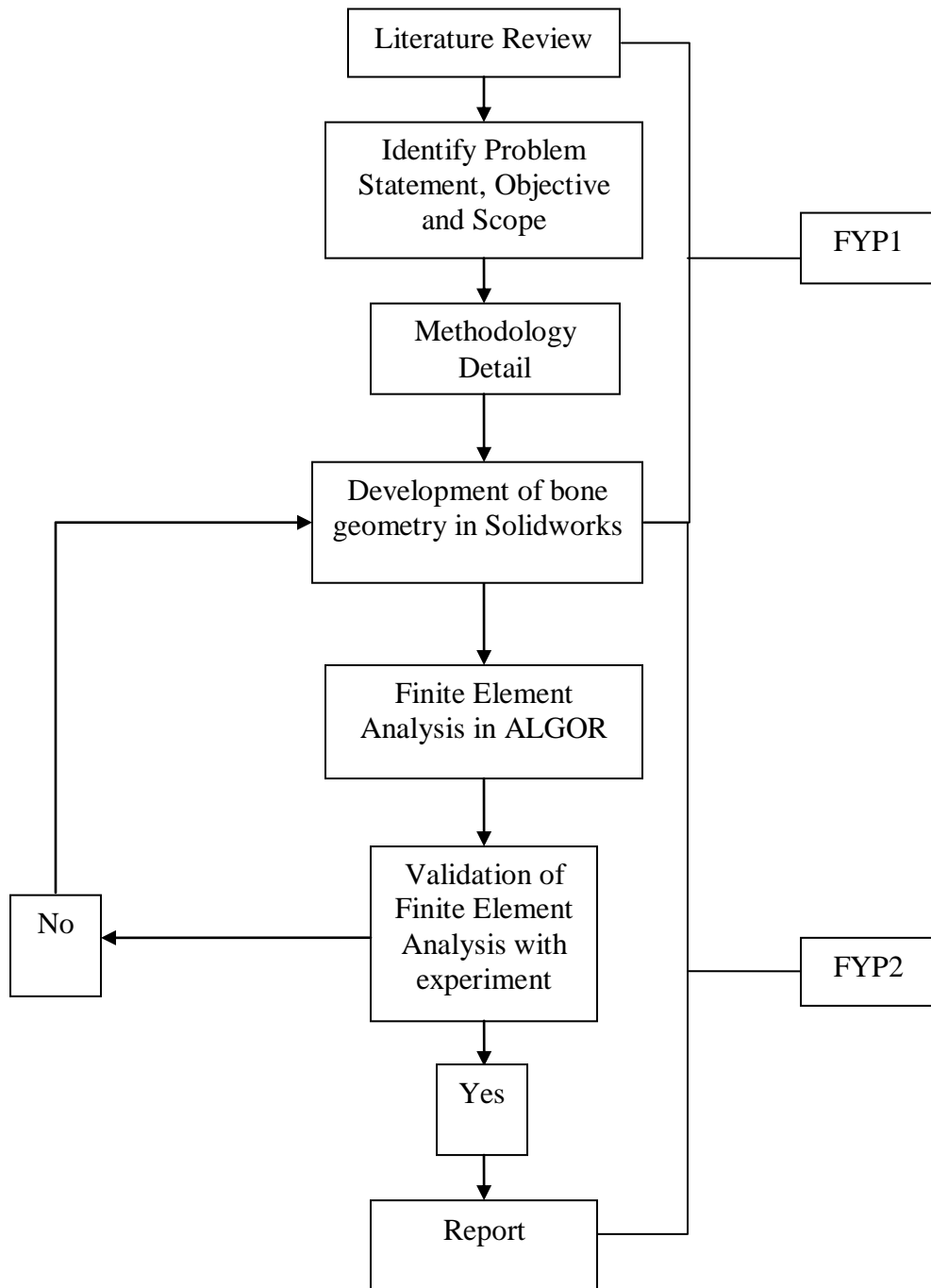


Figure 3.1: Methodology Flow Chart

3.2 THE WHOLE PROJECT FLOW

3.2.1 Literature Review

The project started after receiving the title. Then, the project need to be studied first by doing research for previous study from the published reports and published books. By doing the research, the general information regarding to the title can be gathered. It also helped in designing the suitable flow for this project.

3.2.2 Identifying the Objective, Problem and Project Scope

The objective of this project needed to be determined first by consulting the supervisor. From that, the problem statement and the project scope regarding with the title can be determined. With all the requirement of this project had been gathered, the project can be established.

3.3.3 Methodology Detail

In methodology detail the need to determine the overall step or procedure of this project. This project have three main step which is development of bone geometry in Solidworks, finite element analysis in ALGOR(FEMPRO) and the last step is validated of finite element analysis model with experiment.

3.3.4 Development of Bone Geometry in Solidworks

First step of this project need to develop bone geometry. The software had been decide to used is a Solidworks. The bone geometry is developing according to real size of bone synthetic. Bone porosity size is built from 0.2 to 0.3 mm. This size is according to the researchers that have been done before this. The bone geometry is cut to the simpler model to make it easy opened is ALGOR.

3.3.5 Finite Element Analysis in ALGOR

The software that has been decided to analysis the bone synthetic is an ALGOR (FEMPRO). After bone geometry is successfully developed, the file is saving as IGES and it is import to ALGOR. In ALGOR the model is start to be mesh. Mesh size of the bone geometry is reduced from coarse to fine mesh. For this project the mesh size of bone geometry is need three different mesh size. The mesh size that has been decided for the bone geometry is 100%, 90% and 80% mesh size. After all the three models with different mesh is successfully simulate, the model is select according to the mesh convergence result.

3.3.6 Validation of Finite Element Analysis with Experiment

To complete the project and also to ensure the success of the project, the model must be validated. The validation will be based on the comparison of where the loads are concentrated at the bone between the simulation in ALGOR and the experiment. If the result of the validation is similar to each other the project is success and goes to the report writing. If the validation is not accurate and not similar the process will go back to the development of bone geometry.

3.3.7 Report Writing

The last step of this project is report writing. After all the result is get and the validation of model is successful, report of this project is write and compile all the step start from introduction until the model validation.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

Result of the project is get after all steps in methodology section is done. In this section, all the result is show. Then the result of the simulation will be validated with the experiment. This is important to show that whether result is correct or not.

4.2 DEVELOPMENT OF BONE GEOMETRY IN SOLIDWORKS

Solid works software is used to develop the model of bone. Several size and design of the bone had been modeled. From the research two appropriate bone models have been selected where one is a very complex model and another is the small cut out from the bigger model. The original shape of the bone is a cylinder shape where the original model has 8 mm diameter and 20 mm length and pores size ranging from 0.2 to 0.3 mm. Before make it to cylinder shape, the bone was design in rectangular shape. The pores are created using spline tool and then extrude cut to make the holes on the rectangular shape. Figure 4.1 show the rectangular bone model.

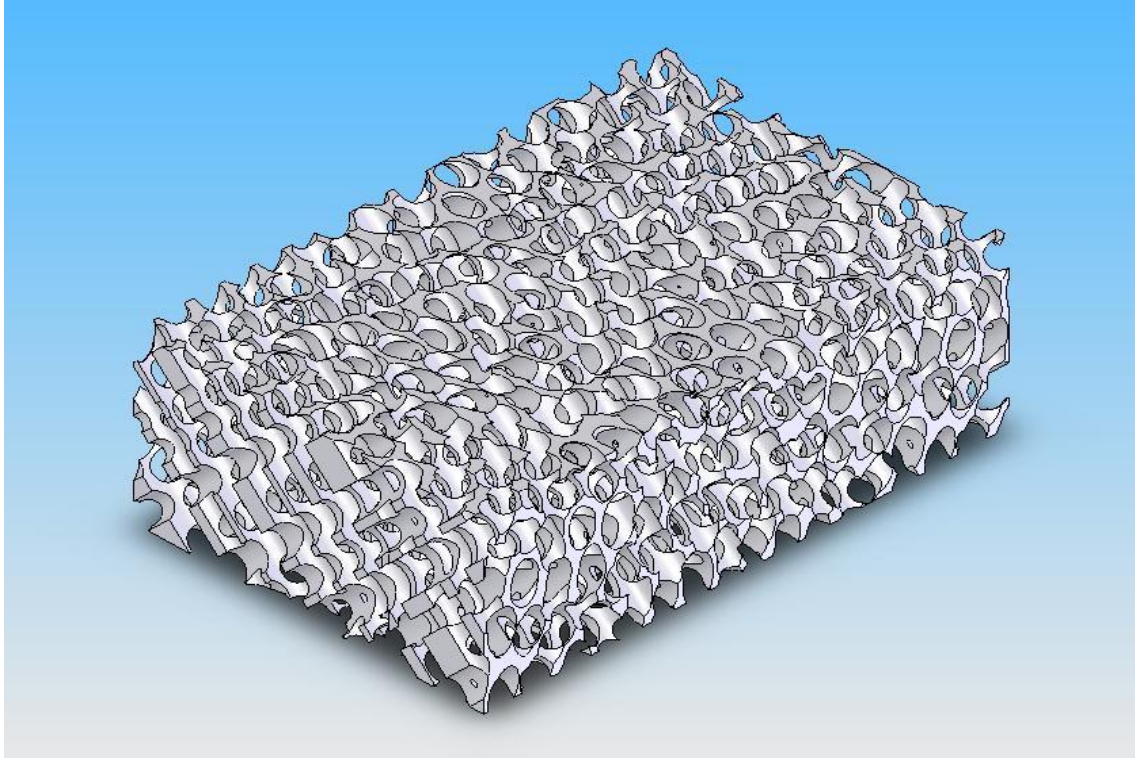


Figure 4.1: Rectangular bone model

After all pores are created of top, front and right plane of the rectangular, the model were extrude cut to cylinder shape. Figure 3.3 shows the complex cylindrical shape of bone. The simpler model of bone that were cut form initial model has 1.6 mm diameter and 2 mm length. Figure 3.4 shows the simpler cut out model from the initial model. This is important because when to simulate the model in ALGOR the simpler model is good then use the complex model. The software is easy to understand the simpler model better than the complex model. When finished with the model is then to be saved as an IGES file where it can be opened using ALGOR.

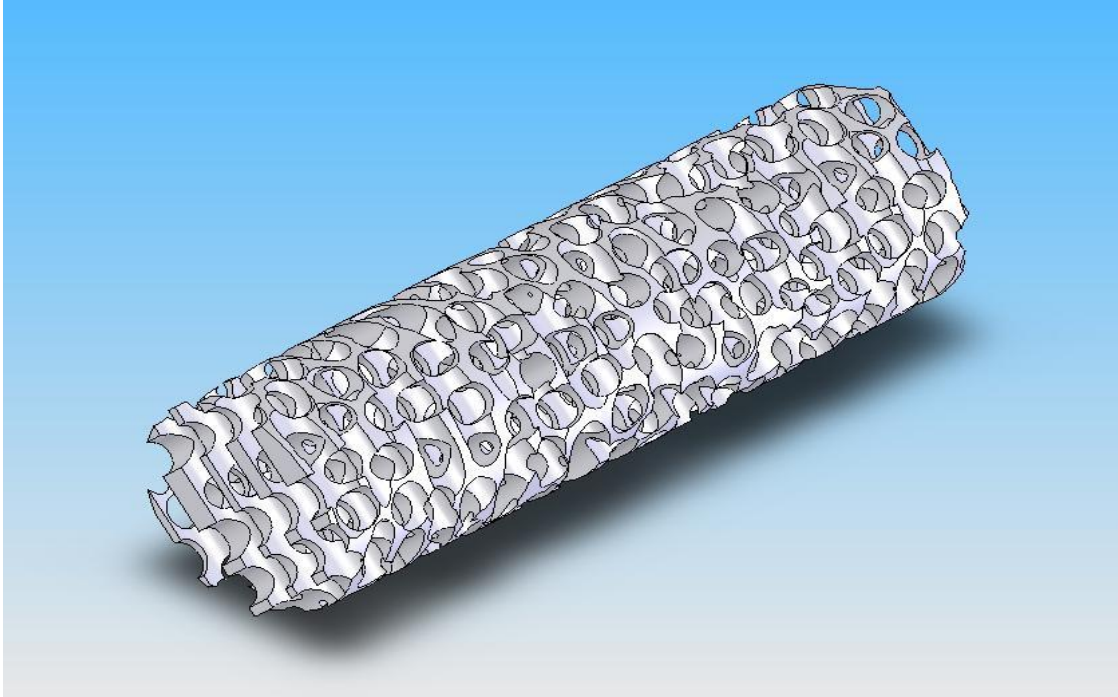


Figure 4.2: Cylindrical bone models

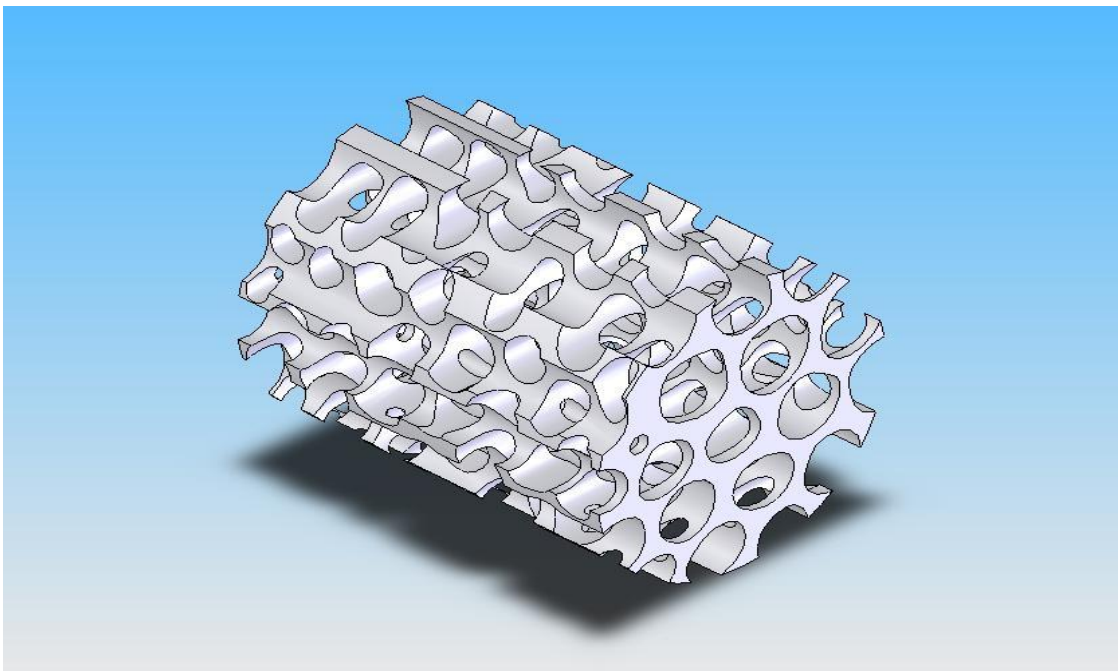


Figure 4.3: Cylindrical bone model with reduced size

4.3 FINITE ELEMENT ANALYSIS

The finite element analysis of the bone is done using Algor V22 software. To begin the analysis the bone that has been modeled using CAD environment need to be opened in the Algor CAD environment. This is done by opening the IGES file of the bone model. After successfully opened the next process is to set the unit of the whole analysis. The Units are done using custom set up the main parameter is the Force in Newton (N), the length in millimeters (mm) and time in Seconds (s).

The next process is to mesh the bone model. This is a crucial part as it plays a major role in getting an accurate result and to verify the model. To find the optimum mesh size the model need to be mesh in different mesh size from course mesh to a finer mesh. Basically the result obtained will be accurate when a finer mesh is done, but as to fine the optimum mesh size for this analysis the largest mesh size is 100 % then decrease gradually. It is advised to not to take the mesh size to close as the results doest not vary much from each other. As for in this analysis, the mesh size will be 100%, 90% and 80. Start by setting the required mesh size and mesh the model. When finish the meshed model will then open in a FEA editor environment. Figure 4.4, 4.5 and 4.6 below show the bone model with the 100%, 90% and 80% mesh respectively.

ALGOR.

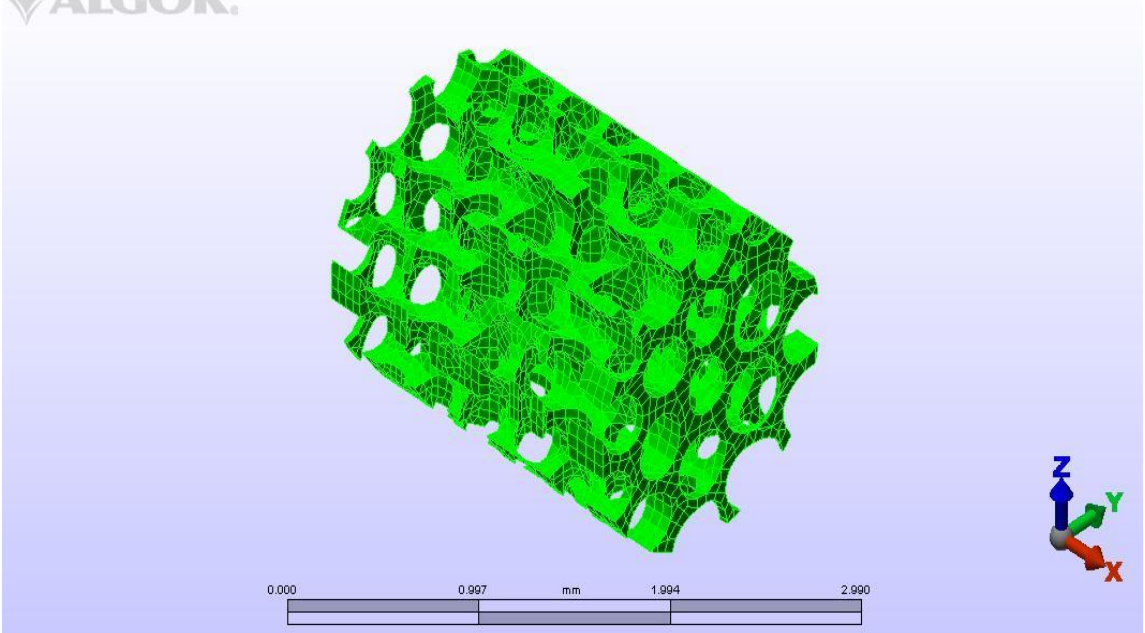


Figure 4.4: Bone model with 100% mesh size

ALGOR.

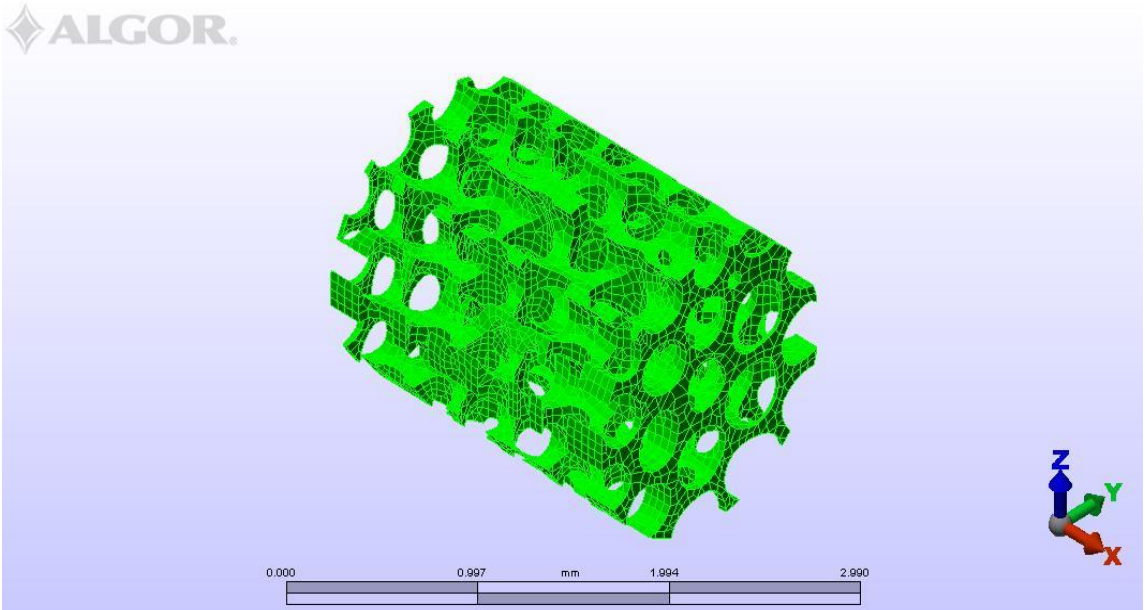


Figure 4.5: Bone model with 90% mesh size

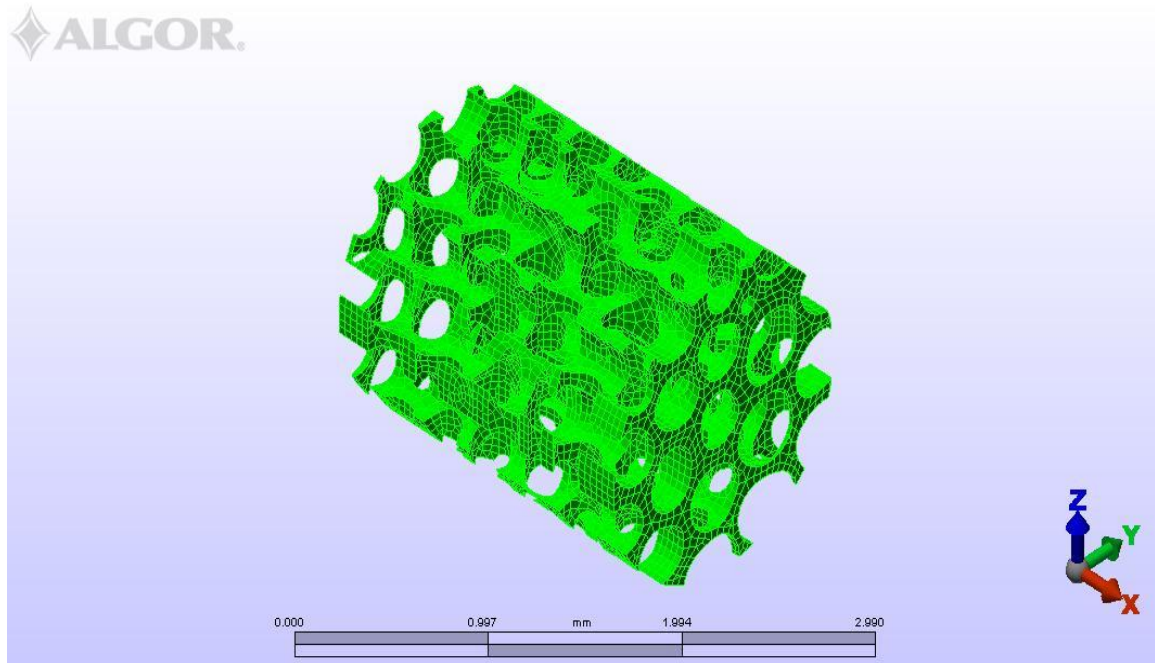


Figure 4.6: Bone model with 80% mesh size

The analysis type was used non linear analysis- Mechanical Event Simulation (MES) combine with non linear material. MES combine kinematic, rigid and flexible body dynamics and non linear stress analysis capabilities. As a result MES can simultaneous analyze mechanical event involving large deformation nonlinear material properties; kinematic motion and forces caused by the motion and then predict the resulting stresses. Some of the main advantages of MES are the need to make fewer assumptions. With MES, there is no need for elaborate hand calculation, interpretation of results or experiments to determine equivalent loading.

The analysis parameter has to be modified in order to simulate the impact of the bone model. The duration is set to be 1 second and the capture rate is 20. This mean the duration of force applied to the bone is in 1 second. Element type is to be determined next, where the element type for the bone model can be considered as brick. In order to

make the software understands the property of the material, element definition comes into consideration. Element definition for MES with nonlinear analysis includes elastic and plastic characteristic of the material besides few specific element definitions.

To analyze human bone the element definition chosen is the Drucker-Prager material model. This model is similar to the standard elastic/plastic material model except that the assumption of a yielding function is different. The Drucker-Prager is same properties with the granular materials. Granular material is a material that has brittle properties which is it is easy to crack such a rock and concrete. So it is use because of bone is considered as a brittle material which is similar to the granular materials.

Next is to define Drucker-Prager material properties. Properties that need to set are the material mass density, Young's Modulus, Poisson's Ratio, damping and the yield function β . The modulus of elasticity or Young's Modulus is the slope of the stress versus strain curve of a material until the proportionality limit, or yield stress. All the properties for this analysis is obtained from the researchers done earlier.

Table 4.1: Material Properties Used for Drucker-Prager

Properties	Value
Material mass density	16 g/cm ³
Young's modulus	14.8 GPa
Poisson's Ratio	0.3
Damping coefficient C	0.00115 s
Yield Function β	0.057 MPa

After all the parameters are set the next step is to set the boundary condition and loads. For the impact test bone is clamp at the end so in ALGOR bone is fixed at the end in all direction at plane Z axis. Next is to set the nodal forces by selecting some nodes of the bone model at the X-Y plane and apply nodal forces in positive Y direction to simulate the impact. Here the force applied for each node must be calculated separately, total load divided by number of nodes, as the number of nodes for each mesh condition will differs. The value of the force that can be applied to the impact test is taken from the values of pendulum energy that be used in experiment done by other. Table 4.2 show the pendulum energy that be used in experiment.

Table 4.2: Pendulum energy and pendulum length

Pendulum energy, (J)	Pendulum length, (mm)
1	390
2.75	390
10.8	390

In the simulation impact of bone in ALGOR the values of force that is used which the pendulum energy with 10.8 J. To convert the unit Joule to Newton the pendulum energy must be divided by the pendulum length. 10.8 J divided by 0.390 m of pendulum length, the force that can be applied is 27.690 N.

For the 100% mesh size the total force applied is 0.503 N which is 27.690 N divides by 55 numbers of nodes. The illustration for the boundary condition and load are shown in figure 4.7.

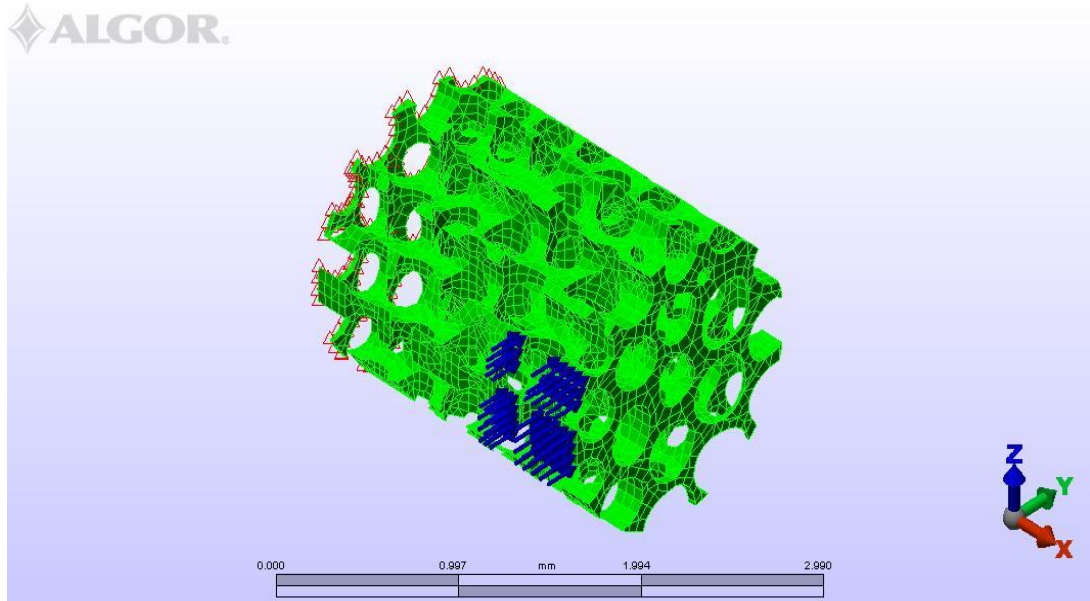


Figure 4.7: Nodal and Boundary condition applied to 100% mesh size model

For the 90% the total force applied is 0.390 N which is 27.690 N divides by 71 numbers of nodes. The illustration for the boundary condition and load are shown in figure 4.8.

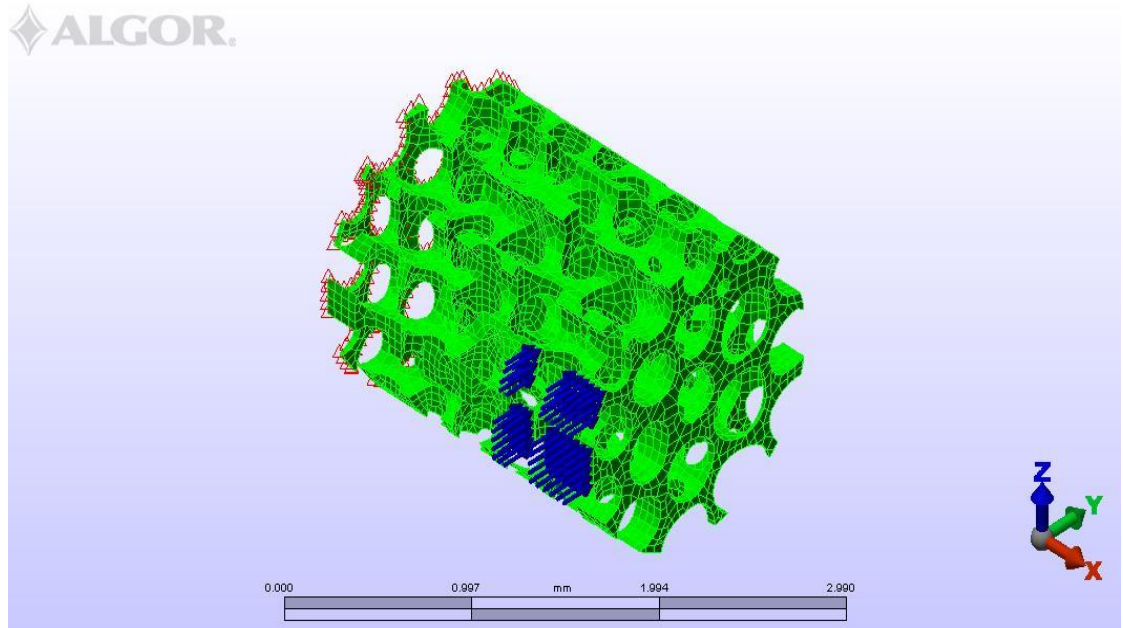


Figure 4.8: Nodal and Boundary condition applied to 90% mesh size model

For the 80% the total force applied is 0.142 N which is 27.690 N divides by 195 numbers of nodes. The illustration for the boundary condition and load are shown in figure 4.9.

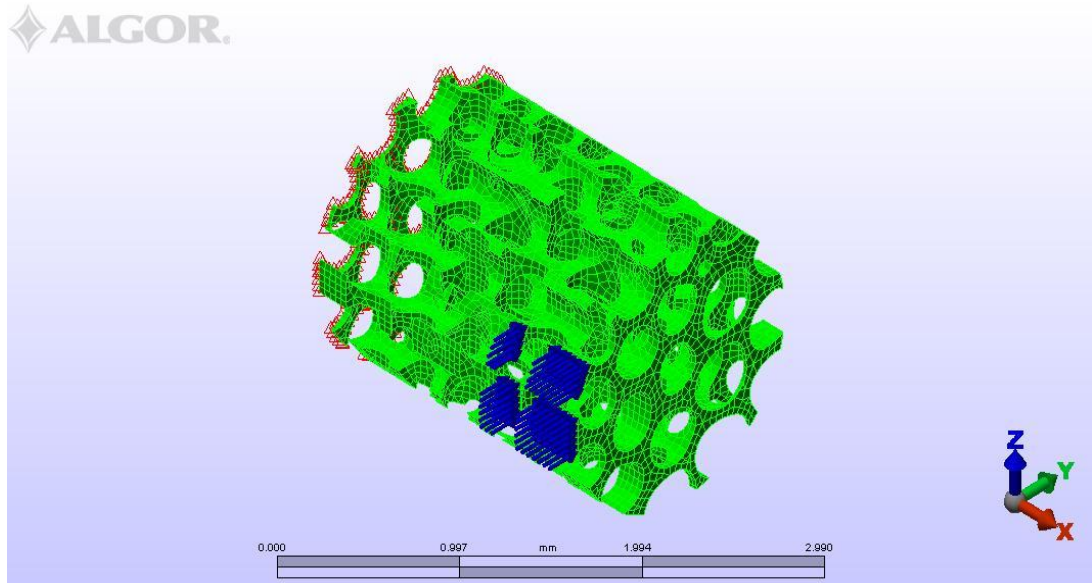


Figure 4.9: Nodal and Boundary condition applied to 80% mesh size model

After all the required parameters are set, the model is now ready to be done analysis. To perform analysis is rather easy, just click the perform analysis button and the rest will be done by the software it self. The analysis is done on the 100%, 90% and 80% mesh sized model to find out the optimum mesh size.

4.4 MESH CONVERGENCE

Mesh convergences is done to select which model should be analyze. The model will be selected by comparing the value of the Stress Von Mises. The Stress Von Mises is an overall stress in all direction. The bone model that wants to be analyzed is a 3-D model, by using the Stress Von Mises, will get all the stress value for all direction. If the value of the Stress Von Mises is near or convergence to each other the model will be select by the finer of the mesh size.

4.5 COMPUTED RESULT WITH 100% MESH SIZE

The maximum Stress Von Mises for bone model with 100% mesh size is 0.1193567 (N/mm²). Figure 4.10 shows the Stress Von Mises contour for model with 100% mesh size. Figure 4.11 shows the Stress tensor Y-Y contour for model with 100% mesh size. Figure 4.12 shows the Stress tensor Y-Z for model with 100% mesh size.

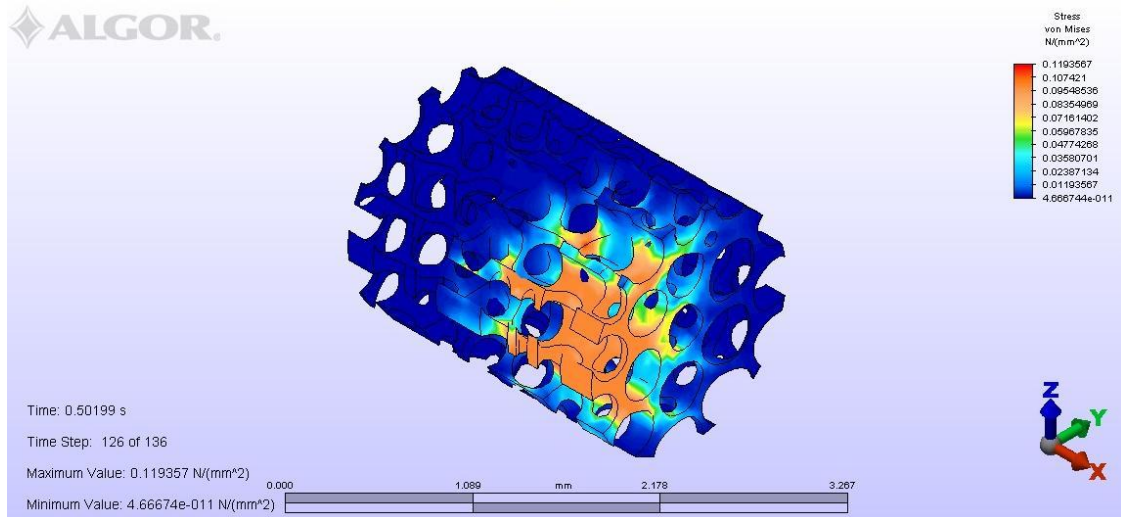


Figure 4.10: Stress Von Mises contour for bone model with 100% mesh model

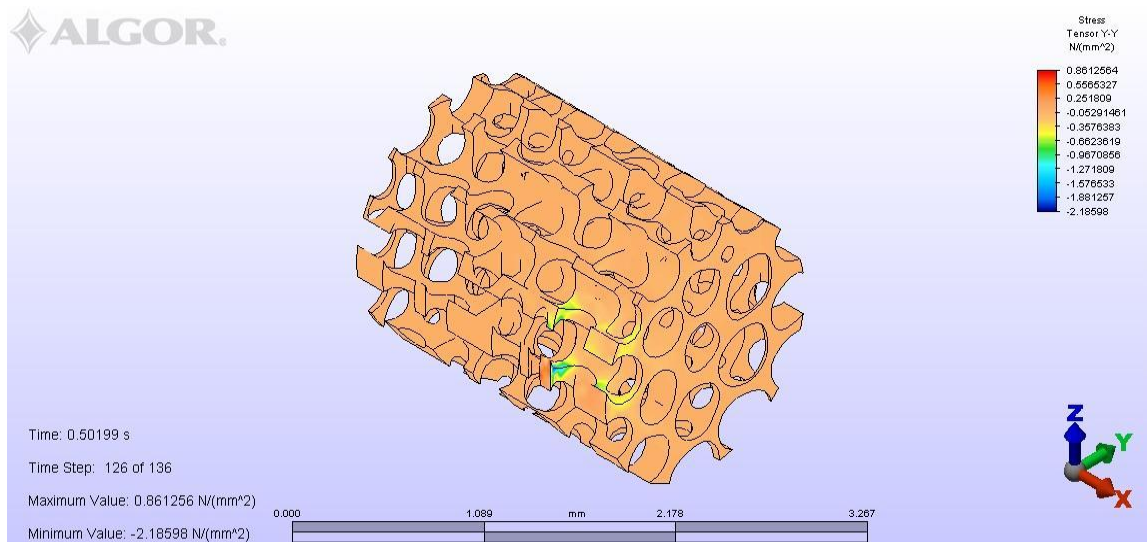


Figure 4.11: Stress tensor Y-Y contour for bone model with 100% mesh size

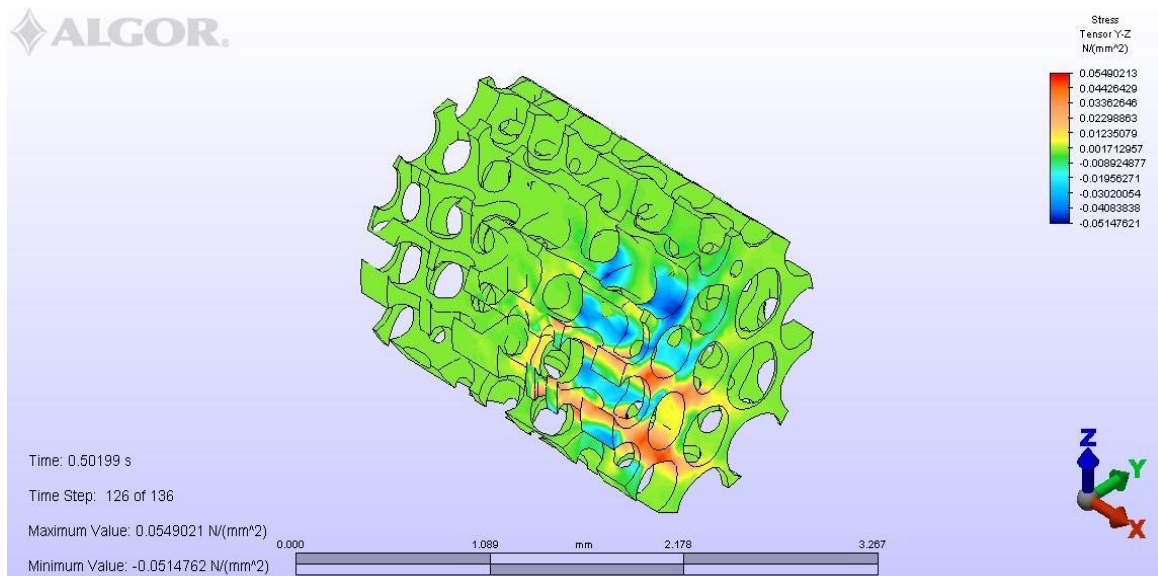


Figure 4.12: Stress tensor Y-Z contour for bone model with 100% mesh size

4.6 COMPUTED RESULT WITH 90% MESH SIZE

The maximum Stress Von Mises for bone model with 90% mesh size is 0.1184711 (N/mm²). Figure 4.13 shows the Stress Von Mises contour for model with 90% mesh size. Figure 4.14 shows the Stress tensor Y-Y contour for model with 90% mesh size. Figure 4.15 shows the Stress tensor Y-Z for model with 90% mesh size.

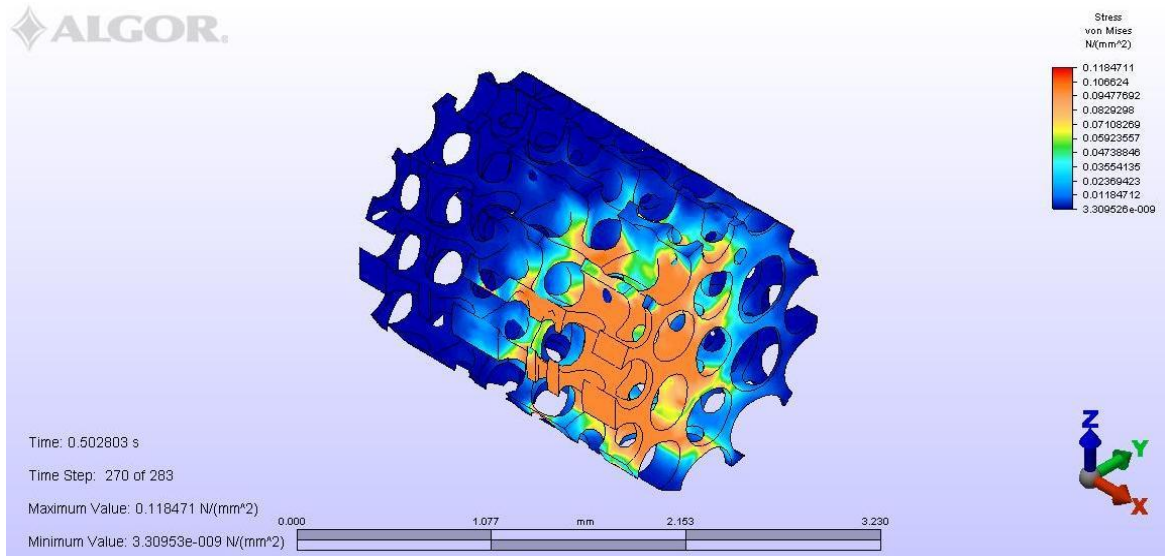


Figure 4.13: Stress Von Mises contour for bone model with 90% mesh size

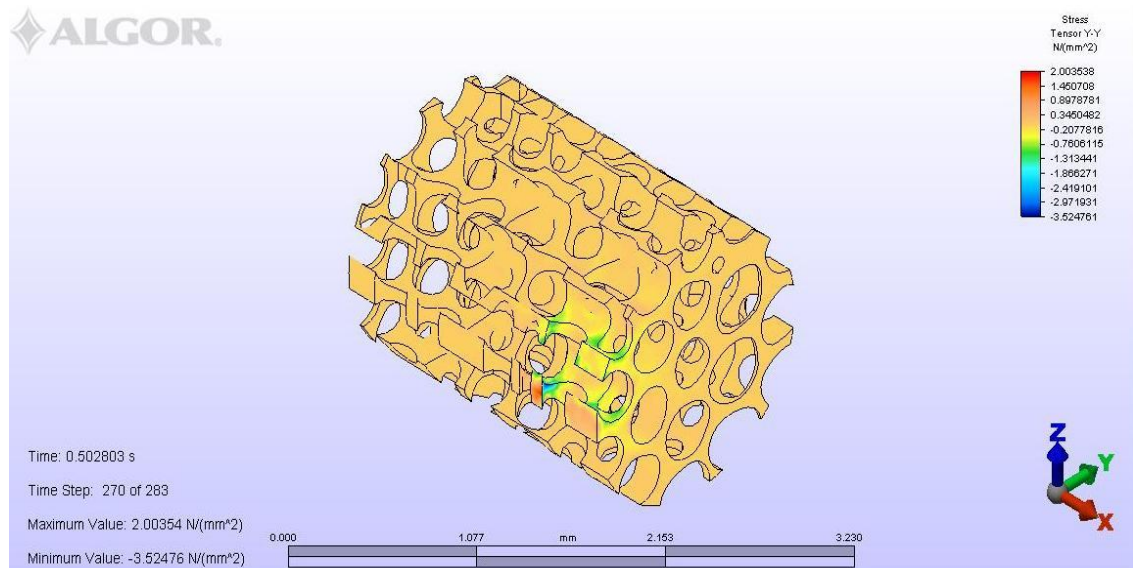


Figure 4.14: Stress Tensor Y-Y contour for bone model with 90% mesh size

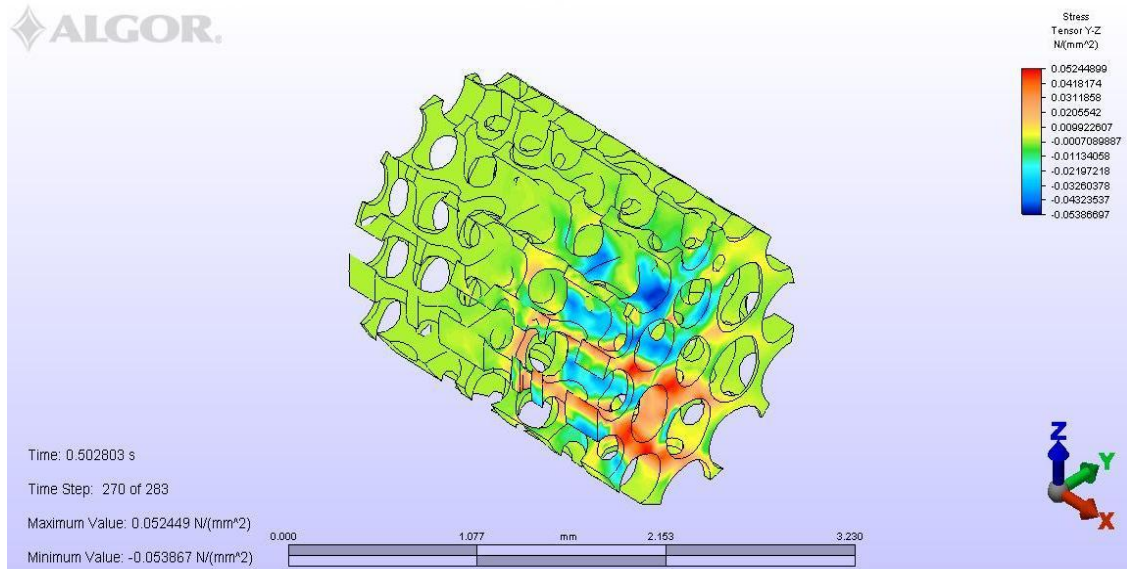


Figure 4.15: Stress Tensor Y-Z contour for bone model with 90% mesh size

4.7 COMPUTED RESULT WITH 80% MESH SIZE

For the bone model with 80% mesh size, the simulation cannot be done successfully. This is because the computer that be used not sufficient to analyze the bone with increase number of element. When the mesh size becomes small the number of element will increase. Figure below show several result for model with mesh size 80%.

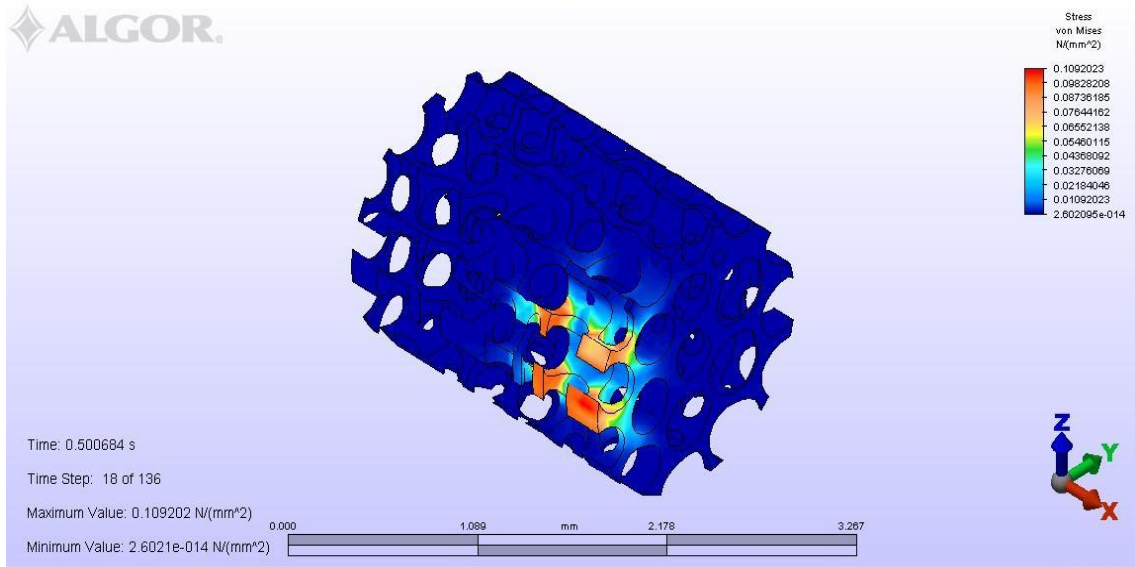


Figure 4.16: Stress Von Mises contour for bone model with 80% mesh size

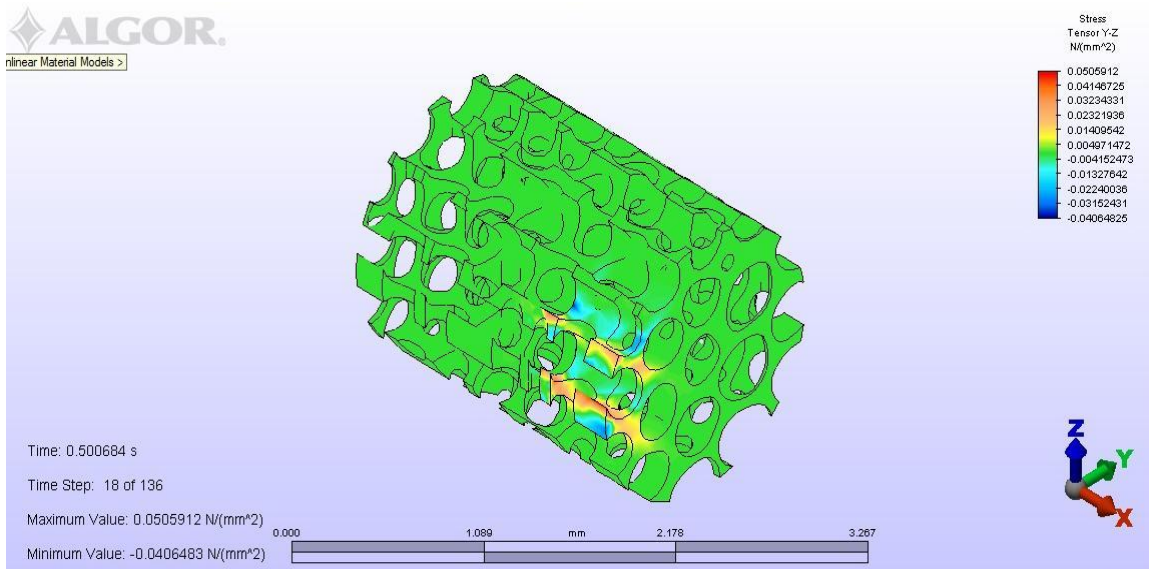


Figure 4.17: Stress Tensor Y-Z contour for bone model with 80% mesh size

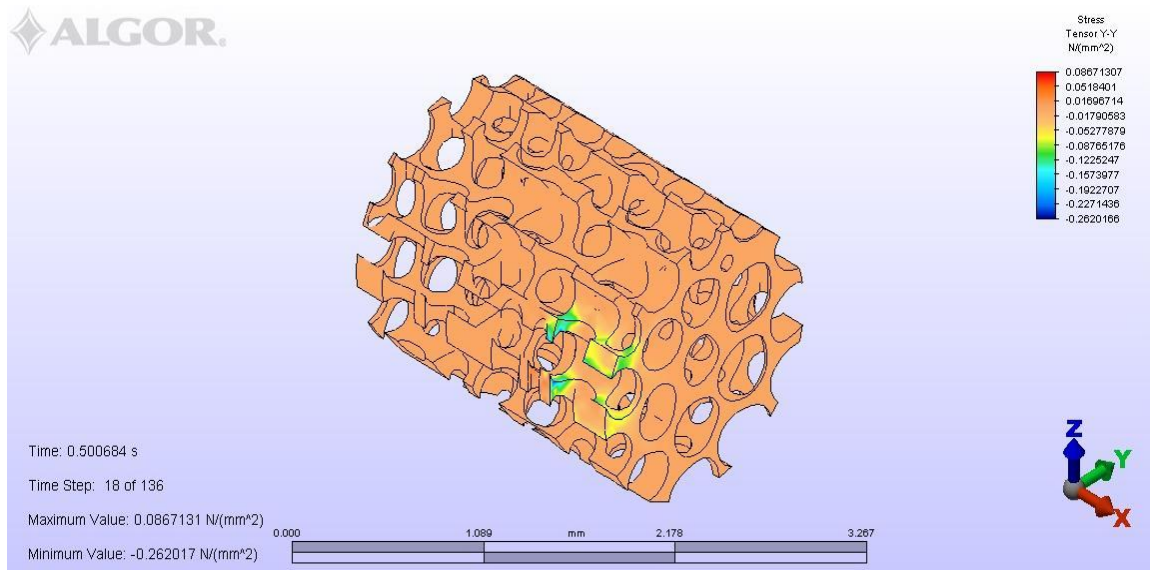


Figure 4.18: Stress Tensor Y-Y contour for bone model with 80% mesh size

4.8 CONVERGENCY ANALYSIS

Table 4.3: Result for the mesh convergence

Mesh size	Number of element	Stress Von Mises (N/mm ²)
100%	19143	0.1193567 (N/mm ²)
90%	24364	0.1184711 (N/mm ²)
80%	29595	Unable to complete analysis

Based on the table result above, the bone model with 90% mesh size is chosen to be the model for actual result and validated with the model done in an experiment. With references to all the computed result extracted from the analysis, Stress Von Mises was found to be suitable parameter for convergences analysis. Consequently, Von Mises stress was used as a basic of comparison to decide mesh convergence.

Table 4.3 summaries Von Mises stress computed for three different models. It is however, to be noted that analysis of the last model which has very fine mesh was unable to complete due to limitation of computer power. The number of the element become increase for the finer mesh size and the model become more complex, the computer power not sufficient to simulate the complex model. Fortunately, solution converges at model with 90% mesh size.

4.9 BONE RESPONSE

In order to investigate the bone response under impact load, the stress-strain curve should be established. Fro that graph, it will show the bone response. The result obtained from the bone model with 90% mesh size is used to plot the stress-strain curve. All value of the Stress Von Mises and Strain Von Mises is referred at the node number 5622. The node number 5622 is used because the maximum Stress Von Mises value occurs at this node. Table 4.4 and table 4.5 show, the Stress Von Mises and Strain Von Mises for node 5622.

Table 4.4: Stress Von Mises Value for Node 5622

Node	Time (s)	Stress Von Mises (N/mm²)
5622	0.50063	0.00071256
5622	0.50111	0.00827755
5622	0.50137	0.01978549
5622	0.50157	0.03316974
5622	0.50177	0.05134916
5622	0.50195	0.07334492

5622	0.50207	0.07671131
5622	0.50218	0.08213745
5622	0.50230	0.08818606
5622	0.50242	0.09482407
5622	0.50253	0.10174977
5622	0.50265	0.10890672
5622	0.50277	0.11646206

Table 4.5: Strain Von Mises values for Node 5622

Node	Time (s)	Strain Von Mises (N/mm²)
5622	0.50063	0.00000747479
5622	0.50111	0.00009974635
5622	0.50137	0.00018871886
5622	0.50157	0.00027403879
5622	0.50177	0.00034061582
5622	0.50195	0.00040985217
5622	0.50207	0.00048206460
5622	0.50218	0.00056333354
5622	0.50230	0.00063917855
5622	0.50242	0.00068788352
5622	0.50253	0.00073544852
5622	0.50265	0.00078587185
5622	0.50277	0.00080132423

The graph of the Stress Von Mises versus time is in figure 4.19. The graph of the Strain Von Mises versus time is in figure 4.20 and the graph of the Stress-Strain is in figure 4.21.

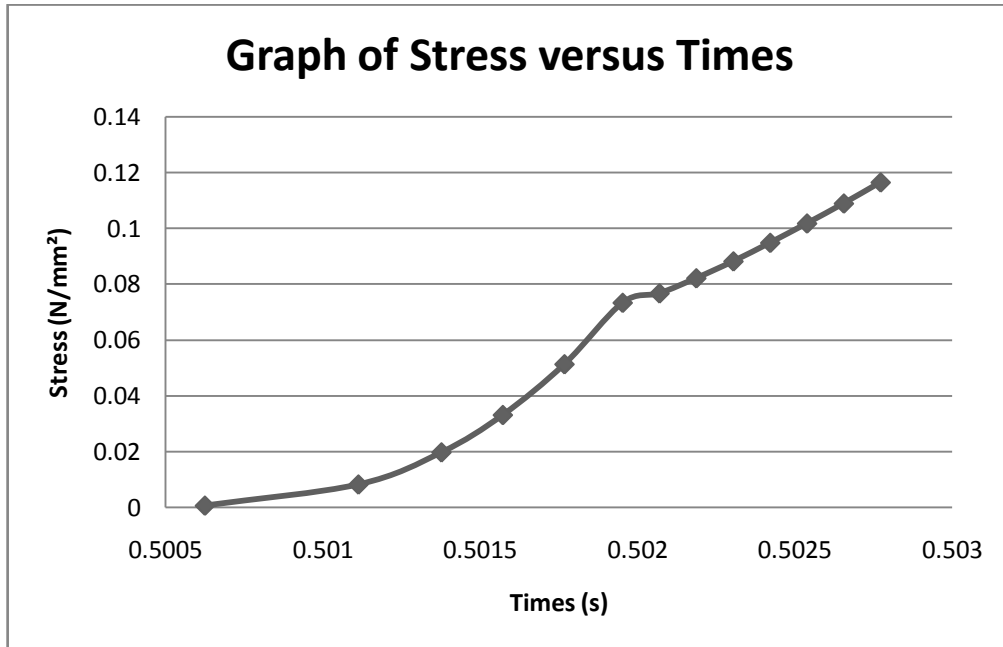


Figure 4.19: Graph of Stress versus Times of node 5622

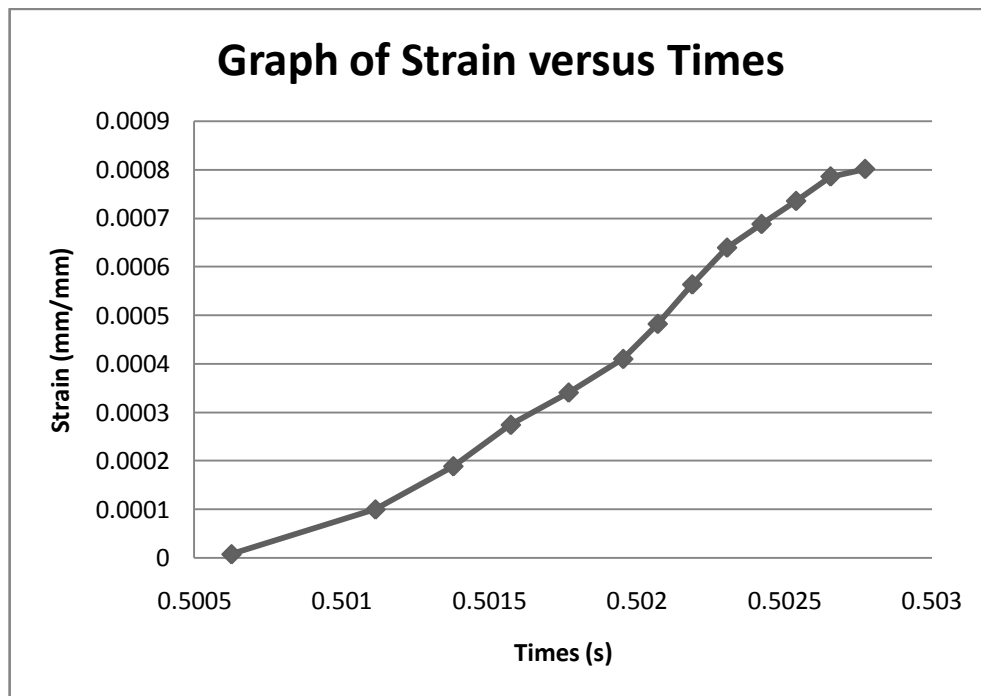


Figure 4.20: Graph of Strain versus Times of node 6522

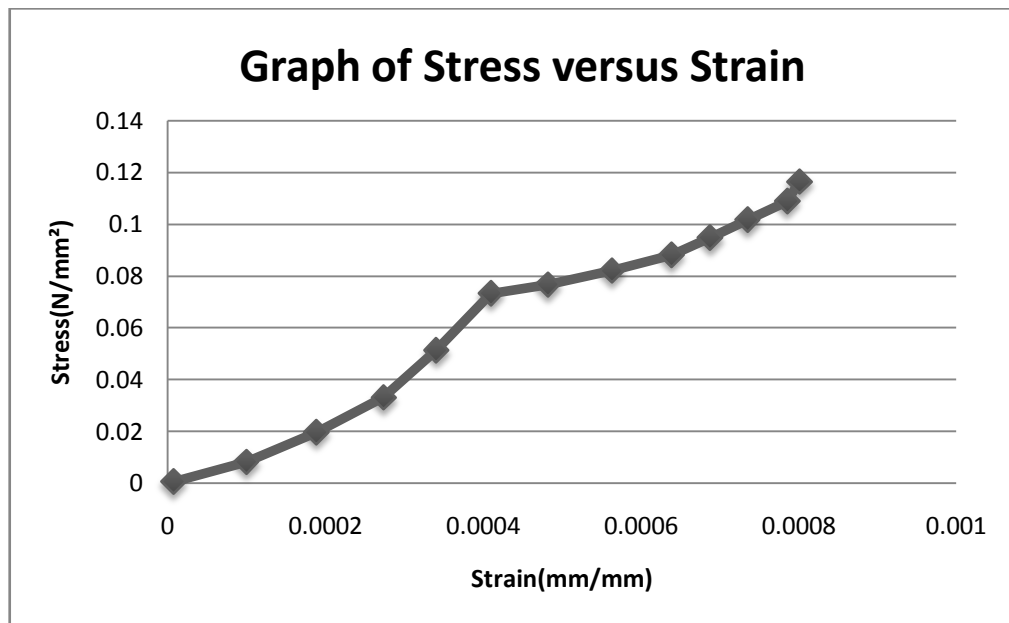


Figure 4.21: Graph of Stress-Strain

From the graph of stress-strain, it shows that how the bone response under impact loads. The graph show that when the stress value increases the strain value also increases. The trend as a whole follows non-linear relationship, indicating that the behavior of the bone response is more to the brittle nature.

4.10 MODEL VALIDATION

The bone model was qualitatively validated by comparing the simulation location of stress of impact load concentration with experimental result. Figure 4.22 and 4.23 show simulated and experimental result respectively. The results were found to be in a good agreement as predicted stress concentration was observed almost the same as the experimental one.

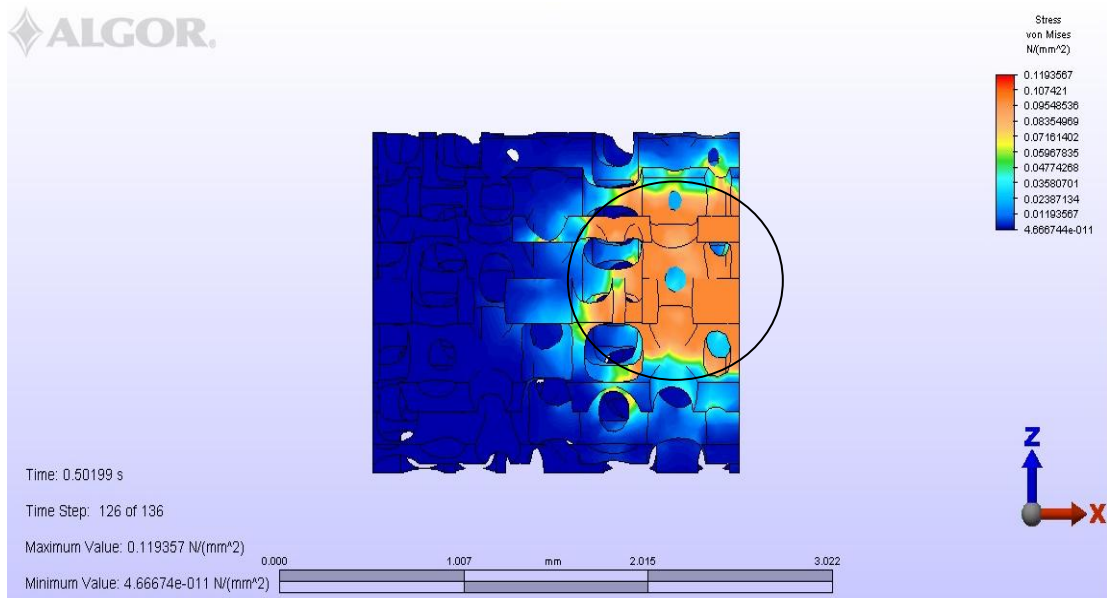


Figure 4.22: Load concentrated in FEA model

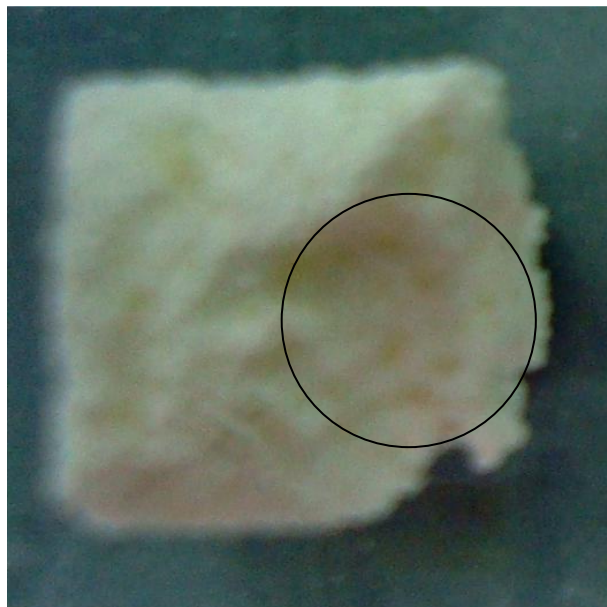


Figure 4.23: Load concentrated in impact experiment

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

The study of the response of human bone synthetic under impact load using Finite Element Method (FEM) has provided a very good knowledge to understand and to determine the characteristic of human bone under impact load. Besides that, it also helped in full utilizations of the finite element packages and ways to overcome a certain limitation of the analysis. The followings are the few conclusions drawn out from the research of the project.

- I. Computational model of bone synthetic has been successfully developed using Finite Element Method (FEM) to predict impact response. This project had developed three model of bone using finite element. The three models had a difference mesh size which 100%, 90% and 80% mesh size. Mesh convergence is used to select which one the best or accurate bone model. The bone model with mesh size 90% is the choose model would be to analyze.
- II. The modal has been qualitatively validated with the impact experiment. For this project, the bone model is validated only on the qualitative. The bone model with mesh size 90% is validated with the bone synthetic in impact experiment by comparing the contour of the load is concentrated. According to the qualitative validated, the load are concentrated in simulation in similarly to the load are concentrated to the bone in the impact experiment.
- III. Stress strain curve that indicates bone response under impact load has been established. From the stress strain graph, it show that the how bone response under impact load. The graph of stress strain that been established is the

generally non linear. It shown that the behavior of the bone response to the brittle nature.

5.2 RECOMMENDATION

This is a few of the recommendation to make this study become more smooth, easy and accurate:

- I. Use more advance software that have a properties of bone. This study were used only simulation in ALGOR. The properties of Ducker-Prager are used to analysis the computational bone model. The parameter of Ducker-Prager is getting by the literature that has been done before this study. In the future more advance software that has properties of bone should be used. The result that will get is being more accurate.
- II. Use a quantitative validation of a model with the experiment. These study only validated bone models with the experiment on the qualitative by comparing the contour of the load are concentrated. For the future study should validate the quantitative validation of a model with the experiment such as comparing the stress or strain value for the simulation and the experiment.
- III. The hardware or computer used must have more powerful specification. The computer that be used must have a good sufficient to analysis the complex model. When doing the mesh convergence the model have a difference mesh size. The number of element is increase when the size of mesh becomes smaller. The model will be more complex. To do the smooth simulation, the hardware or computer must have more power specification.

REFERENCES

- Arbenz, P.Prof. Dr., Lenthe G. H. van Prof. Dr., Müller, R. Prof. Dr., Sala M. Dr, (2006), *μ - Finite Element Analysis of Human Bone Structures*. <http://ercim-news.ercim.org/content/view/406/641/>. Ercim Inc
- Enderle, John D., Bronzino, Joseph D.,and Blanchard, Susan M., 2005. *Introduction To Biomedical Engineering*. Elsevier Inc.
- Follet H., Martinais R., Mbodj C., Fuller G.1, and Delmas P.1, (2006) *Intrinsic Mechanical Properties of Human Vertebral Trabecular Bone: Influence of the Finite Element Model*. Inserm
- Gao, Huajian, Ji, Baohua, *Nanoscale Mechanical Properties In Bone And Dentin*. Paper presented at the Summer Bioengineering Conference, Sonesta Beach Resort in Key Biscayne, Florida. June 2003.
- Gedney, Richard T., (2007). *Trends in Mechanical Testing*, www.admet.com. Admet Inc.
- Hollister, Scott J. PHD, (2007). BME/ME 456 *Biomechanics: Bone Structure*, <http://www.engin.umich.edu/class/bme456/bonestructure/bonestructure.htm>.
- Hutton, David V., (2004). *Fundamental of Finite Element Analysis*. Mc Graw Hill Inc.
- Kalpakjian, Serope, and Schmid, Steven, (2005), *Manufacturing, Engineering and Technology*, 5th Edition. Pearson Education.
- Materials Testing. (2008). *Encyclopædia Britannica. Encyclopædia Britannica 2007 Ultimate Reference Suite*. Chicago: Encyclopædia Britannica.
- Meyers, Marc A., and Chawla, Krishan Kumar, 1999. *Mechanical Behaviors of Materials*. Prentice Hall Inc.

Olszta, Matthew J., Cheng, Xingguo, Jee, Sang Soo, Kumar, Rajendra, Kim, Yi-Yeoun, Kaufman, Michael J., Douglas, Elliot P., Gower, Laurie B., (2007). *Bone structure and formation: A new perspective*. Journal of Materials Science and Engineering.

Sala, Marzio. (2006), *Multi-level μ -Finite Element Analysis of Human Bone Structures*, ETH Zurich.

Smith, William F., and Hashemi, Javed, (2006). *Foundation of Material Science And Engineering*. Mc Graw Hill.

Widas, Peter, (1997). *Introduction to Finite Element Analysis*, Virginia Tech Material Science and Engineering,
www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/num/widas/history.html.

APPENDIX A1

GANTT CHART FOR FYP1

Project Activities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	11	12	13	14
Literature review														
Identify problem														
Objective and scopes														
Developing model geometry in Solidwork														
Learning ALGOR														
Proposal Writing														
Presentation preparation														
FYP 1 presentation														

APPENDIX A2

GANTT CHART FOR FYP2

Project Activities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14
Developing model geometry in Solidwork														
Analysis of impact response in ALGOR														
Model validation with experiment														
Report writing														
Presentation preparation														
FYP 2 presentation														