EXPERIMENTAL & NUMERICAL STUDY ON SHEET METAL LATERAL BENDING WITH BOTH ENDS PINNED

MOHD ADZLAN BIN MOHD AZMI

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

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled “Experimental & Numerical Study on Sheet Metal Lateral Bending With Both Ends Pinned” is written by Mohd Adzlan Bin Mohd Azmi. I 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. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Examiner, Signature

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 with Manufacturing Engineering.

Signature	:	
Name of Supervisor	:	MR. MOHD ZAIDI SIDEK
Position	:	LECTURER
Date	:	22 JUNE 2012

STUDENT'S DECLARATION

I hereby declare that this report entitled "Experimental & Numerical Study on Sheet Metal Lateral Bending With Both Ends Pinned" is the result of my own research except as cited in the references. The report has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature	:	
Name	:	MOHD ADZLAN BIN MOHD AZMI
ID Number	:	ME 08035
Date	:	22 JUNE 2012

DEDICATION

Special dedication to my family members and that always inspire, love and stand beside me, my supervisor, and my beloved friends, my fellow colleagues,
all Faculty of Mechanical lecturers and members.

Thank you so much for your love, care, support and believes in me.

ACKNOWLEDGEMENTS

With the name of ALLAH, the Most Merciful and the Most Beneficial. First and foremost, many praises for ALLAH and whole-heart grateful to the Great Creator as I have finished my final year project.

Starting with a deep sense of gratitude, I would like to express my deepest appreciation to my supervisor Mr. Mohd Zaidi Sidek for his germinal ideas, proficient guidance and continuous encouragement throughout this impressive project. He always consult and acts as my mentor whenever I faced problems. His invaluable advice and sharing of experience indirectly enriches my practical knowledge which are helpful for me to accomplish this project successfully.

Besides that, I acknowledge my sincere indebtedness and gratitude thanks to my family members specially my parents En. Mohd Azmi Bin Che Omar and Puan Sarbani Bte Sulaiman. Nur Athia, Nur Alliya and Aishah for their limitless of caring, spiritual and financial support. I was inspired to attain the goal of the project through their love and endless encouragement. I also want to thank to my colleagues who always brainstorming additional new ideas, suggestion and help me in my project.

Finally I want to thank all the direct and indirect supports helped me completing my Final Year Project in time. May Allah bless these people always.

ABSTRACT

The main aim of this study is to compare the buckling phenomenon of metal strip under both ends pinned setting. Buckling may be demonstrated by pressing the opposite edge of a plate toward one another. In achieving the objective, this research involves design and fabrication of the buckling test rig, run the experiment on test rig under condition both ends are pinned and to compare the experimental result with computer simulation. This thesis describes the methods of designing, fabricating and experiment the lateral bending test equipment. Started with test rig design, it has been developed. using SOLIDWORKS 2011.It is continued with fabrication work in FKM laboratory. Specific for the pinned ends design, V shape feature has been adopted in the design using a rectangular aluminum. The next stage is on experimental work, where the tests have been conducted as the planned test loading. The specimen showed buckling phenomenon and strain has been measured using the strain gauge connected to data logger system. The value of strain from experimental will be compare with strain values from ABAQUS simulation. From this project, we can prove that the experimental result is same with the simulation result.

ABSTRAK

Tujuan utama kajian ini ialah untuk membandingkan fenomena bengkokkan kepingan logam dibawah kedua-dua hujung tersemat. Bengkokkan boleh ditunjukkan dengan menekan tepi plat ke arah bertentangan antara satu sama lain. Dalam mencapai objektif, penyelidikan ini melibatkan mereka bentuk dan rekaan alat penguji pembengkokkan, menjalankan eksperimen pada alat penguji pembengkokkan di dalam keadaan kedua-dua hujung ialah pin dan untuk membandingkan keputusan eksperimen dengan keputusan simulasi komputer. Tesis ini menggambarkan kaedah-kaedah rekaan, mereka dan menguji peralatan ujian lentur sisi. Dimulakan dengan mereka bentuk dan ia dibangunkan dengan mengunakan SOLIDWORKS 2011. Ia diteruskan dengan kerja mereka di dalam makmal FKM. Untuk mereka ciri khusus hujung tersemat, bentuk V telah diambil kira dalam reka bentuk dengan mengunakan aluminium segi empat tepat. Peringkat berikutnya ialah dengan kerja eksperiment, di mana ujian-ujian telah dijalankan dengan ujian pemberat. Spesimen menunjukkan fenomena pembengkokkan dan terikan telah diukur menggunakan tolok terikan yang berhubung dengan sistem analog data. Nilai terikan dari eksperimen akan dibandingkan dengan nilai2 dari simulasi ABAQUS. Daripada projek ini, kami boleh membuktikan bahawa keputusan eksperimen sama dengan hasil simulasi.

TABLE OF CONTENT

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

$\boldsymbol{L_e}$	Effective length
D	Dimension
F	Maximum or critical force
E	Modulus of elasticity
I	Area moment of inertia
L	Unsupported length of strip
K	Strip effective length factor
Pcr	Critical load
R	Resistance
$\Delta\boldsymbol{R}$	Differential resistance
L	Length
t	Thickness
$\mu\varepsilon$	Microstrain

LIST OF ABBREVIATIONS

FEM	Finite Element Method
FEA	Finite Element Analysis
3D	3 Dimensions
SMAW	Shielded Metal Arc Welding
CNC	Computer Numerical Control
m	Meter
mm	Millimeter
DAQ	Data Acquisition
GI	Galvanized iron
GF	Gauge factor

CHAPTER 1

INTRODUCTION

1.1 Project Background

This project focuses on design and fabricate new test rig for buckling test equipment. Then, to run the buckling test rig under pinned- pinned ends condition. Lastly is to compare the result between the experimental and the computer simulation

1.2 Problem Statement

Higher education teaching requires lecturers apply various methods to make the subject easier. In this case, teaching technique is the most important factor. One of the subject is on computer simulation such as finite element method. For this subject, students need to understand the basics engineering mechanics and materials, failure prevention under static and variable loading, and characteristics of the principal types of mechanical elements. With the current practice where student only learn based on following the given tutorial, it is not really practical to make student understand this subject. Hence, there are need changes for current teaching practices because students more prefer hand-on and practical in their learning process. Thus, the new method of teaching is needed to teach subject this subject based on comparison between experiment and simulation analysis. For example, FE course need better teaching technique to make student easily understand their application in solving engineering problem.

Many structures collapse due to lack of support system to counteract lateral loads. Though it is a structural engineer's duty to design the structure for the stability, the contractor must ensure that adequate members have been provided and that the structure is erected as designed. Most of the construction curriculum does not cover lateral stability of the buildings; as a result, students are not aware of its importance. Therefore, there is a need to develop a teaching aid on lateral bending test equipment that will help student to more understanding about buckling.

Therefore, all the construction programs include structures in their curriculum. These construction programs emphasize on in depth study of some of the key issues such as dead loads and live loads, beam, strip design, and so forth and do not focus on entire structural stability. Another primary concern of any structure is to provide sufficient stability to prevent excessive lateral deformation and thus resist collapse. Therefore, along with the vertical loads, every building should be adequately stiffened against horizontal forces like wind and seismic forces. This problem can be investigated by experimental in lab, for example is buckling test, it is a simple engineering problem that always occur in engineering problem.

This innovative capstone serves as a teaching aid on lateral bending of structures that can be used by the professors for the structures classes, by the undergraduate thesis students, and graduate students as a resource. Besides that, it can be tested in lab and Finite Element software can used to develop similar model.

1.3 Project Objective

The main aim of this study is to compare strip buckling with pinned ends by experiment and numerical analysis. To achieve the objectives, the main research activities are :

1. To design and fabricate lateral bending test equipment.
   (Case #2- The both ends are pinned).
2. To run lateral bending test rig under pinned-pinned ends condition.
3. To compare the result between the experimental and the computer simulation.

1.4 Project Scope

This project will focus on the following aspects:

1. To design a lateral bending test equipment by using SOLIDWORKS 2011.
2. To fabricate a lateral bending test equipment.
3. To compare experimental result with theoretical and computer simulations by using ABAQUS. (parameter : buckling behavior and strain).
4. The specimen for experiment test is in both end pinned condition.

1.5 Research Flow

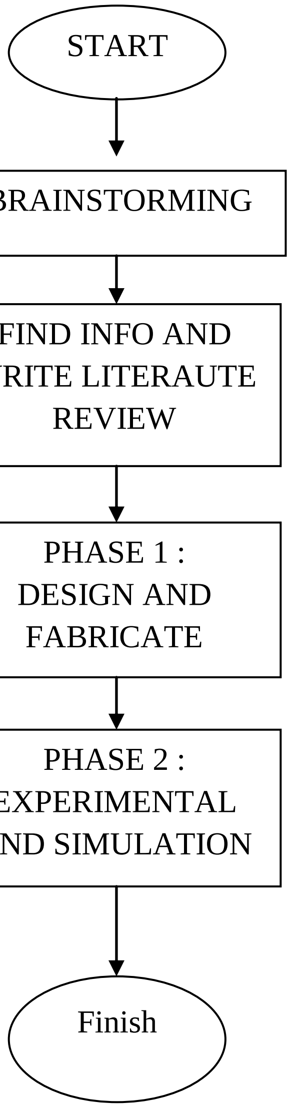


Figure 1.1: Overall Activities Flow Chart

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter provides the detail description literature review done according to the title of “Experimental & Numerical Study on Sheet Metal Lateral Bending With Both Ends Pinned”. Since the aim of this project is to design and fabricate the buckling test equipment, there some are related software, methods and process. Obviously literature review related with definition of buckling phenomenon by run the experimental and simulation. This literature review will give an overview or a brief introduction of the techniques and methods that are suitable to be used in this project.

2.2 Teaching aid

The main idea of this research is to produce a laboratory equipment as a hand-on approach in teaching engineering subject .For certain subject like the finite element method that require a lot of theory and application know ledges, easier for students to understand the topic with more practical approaches. Teaching aid is a tool used by teachers, facilitators, or tutors to help learners improve reading and other skills, illustrate or reinforce a skill, fact, and idea.

In order to make the teaching aid on lateral stability more resourceful, research was conducted to identify ideal ways (best practices) to teach structures to construction students and to study course development strategies for structure’s class. There have been literature available on the course development in general but there have been limited research on the developing contents of structures courses offered by the construction schools. (Chini,1995) conducted a survey of the structures courses offered by ASC member construction schools. The survey was conducted to develop a resource data of text books, audio-visual aids, software used for structures courses. Chini's survey is focused on assessing the need of developing a library of digital images of construction sites, problems, examples that can be used by ASC member schools. He conducted an internet-based survey that asked for additional information about course contents, number of hours of coursework and teaching methodology. Eighty four percent of his respondents believed that there should be emphasis on structures courses in construction schools. On teaching methodology, seventy eight percent of his respondents believed that structures courses should not be taught in the same format as the traditional engineering formats.

He concluded that the emphasis of the structures courses in construction schools should on the construction aspects or implications of design problems and not on the detailed design of individual components. In addition, he stated that lecture format should be combined with experimental work, field trips, audiovisual tapes and guest speakers to make material easy to grasp.

(Williams and Sattineni, 2002) suggested a six-step approach that could help student to understand the concepts. The authors state that developing a problem statement that ties to realistic situations that the students are familiar with, use of simple mathematics, use of what if situations and practice of solving more problems will help students to understand structures better. Though this approach deals with how to teach structures, it narrows down to problem solving and design of details instead of focusing on entire systems. Though the authors do not talk about contents or course development, they conclude that structural courses in construction are taught in the same way as they are in engineering curriculums and that emphasize should be on a basic understanding of the concepts.

Another research by Dr. Burl E. Dishongh (2003) has studied two paradigms for teaching structures and suggested a paradigm that focuses on appreciation of engineering processes and use of sketches, experimental observations, and basic mathematic and physical relationship. Besides that, computer simulation is commonly used by lectures as a teaching aid in engineering subject. It is very important to teach metal forming processes in undergraduate courses of Mechanical Engineering because they are widely used to manufacture a great number of parts to industries like automotive, machine tools, hand tools, surgical instruments, and many others. There is a great difficulty when teaching metal forming due to the large variety of processes, materials and products, which involve specific knowledge strongly related to craftsmen practical experience. It is also in same case in fabrication and design lateral bending equipment.

Computer simulation has become reliable and acceptable in the metal forming industry since the 1980's. Metal forming analysis can be performed in three modeling scales (R. Kopp, M.L. Cao and M.M. Souza, 1987). The first scale is the global modeling, which only predicts process loads or work. Analytical methods are used for this purpose. Local scale analysis is used to estimate the thermo-mechanical variables such as strain, strain rate, and temperature. With the extensive development in computational mechanics, numerical methods have been used as an economical alternative to perform the local modeling. Microscale modeling computes the micro-structural evolution during the forming process. Since global scale analysis is only applicable to simple situations and micro modeling is still incipient and only gives results for specific conditions, local modeling is the most popular approach. Among other methods, the Finite Element Methods (FEM) is widely used in metal forming analysis due to its capabilities to model the complicated shapes of tools and parts in forming processes.

2.3 Buckling

In engineering mechanics, bending (also known as buckling) characterizes the behavior of a slender structural element subjected to a lateral load. A structural element subjected to bending is known as a beam. If the load on a strip is applied through the center of gravity of its cross section, it is called an axial load. A load at any other point in the cross section is known as an eccentric load. A short strip under the action of axial load will fail by direct compression before it buckles, but a long strip loaded in the same manner will fail by buckling (bending), the buckling effect being so large that the effect of the direct load may be neglected. The intermediate-length strip will fail by a combination of direct compressive stress and bending.

Buckling proceeds in manner which may be either:

1. Stable - in which case displacements increase in a controlled fashion as loads are increased. The structure's ability to sustain loads is maintained, or

2. Unstable - in which case deformations increase instantaneously, the load carrying capacity nose- dives and the structure collapses catastrophically.

3. Neutral equilibrium - is also a theoretical possibility during buckling - this is characterized by deformation increase without change in load.

Buckling of structural members has been investigated for over two centuries since the original work of Euler in the 18th century. In recent time, thin-walled members have been developed which use increasingly thinner material, now down to as low as 0.42mm thickness in some cold-formed sections used for residential construction. Section of this type can undergo sectional instability as well as overall modes of buckling.

Buckling has become more of a problem in recent years since the use of high strength material requires less material for load support structures and components have become generally more slender and buckle-prone. This trend has continued throughout technological history i.e. the dangers associated with over-slender build were tragically driven home by the collapse o the Tacoma Narrows road bridge over the Puget Sound in 1940. Although this failure was apparently due to wind-structure aerodynamic coupling rather than buckling.

Buckling can be easily found in strip structure deformation when the strip buckle on the its critical point which it cannot return to original shape and have a high possibility to crack. The problem of bending of beams probably occurs more often than any other loading problem in mechanical design. Shafts, axles, cranks, levers, springs, brackets, and wheels, as well as many other elements, must often be treated as beams in the design and analysis of mechanical structures and systems. (Shigley,2006).

Buckling is a failure mode characterized by a sudden failure of a structural member subjected to high compressive stresses, where the actual compressive stress is less than the ultimate compressive stresses at the point of failure that the material is capable of withstanding. A short strip will fail under the action of an axial load by direct compression before it buckles, but a slender strip will fail by buckling in the same manner. When a compression member becomes longer, the role of the geometry and stiffness (Young's modulus) becomes more and more important. When axial load applies at the strip, the critical load for a slender strip can be determined by the Euler Buckling Load,

$$\boxed{P_{cr} = \frac{C\pi^2 El}{l^2}}$$

Where are:

a) P_{cr} = Critical load
b) C = Constant ends condition
c) E = Young Modulus
d) l = Inertia moments

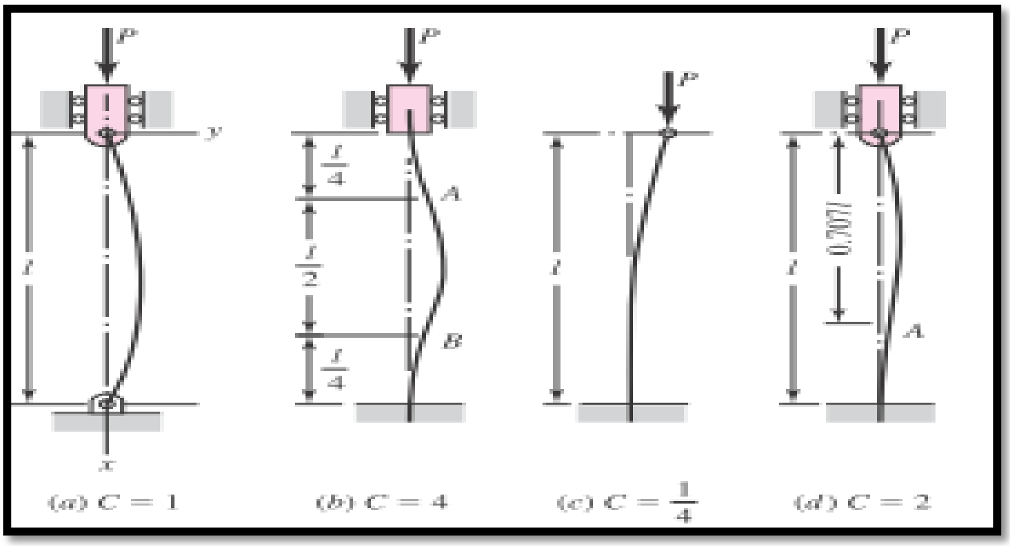


Figure 2.1: Ends Condition of Strip Buckling

Source : Shigley (2006)

Where are,

a) Both ends rounded or pivoted;
b) Both ends fixed;
c) One end free and one end fixed;
d) One end rounded and pivoted, and one end fixed.

In the actual design of a member that functions as a strip, the designer will be aware of the end conditions shown in Fig. 2.1, and will endeavor to configure the ends, using bolts, welds, or pins, for example, so as to achieve the required ideal end conditions. In spite of these precautions, the result, following manufacture, is likely to contain defects such as initial crookedness or load eccentricities. The existence of such defects and the methods of accounting for them will usually involve a factor-of-safety approach or a stochastic analysis. These methods work well for long strip and for simple compression members.

Table 2.1: Ends Condition of Strip Buckling

	End- Condition Constant C		
Strip End Condition	**Theoretical Value**	**Conservative Value**	**Recommended Value***
Fixed-Free	1/4	1/4	1/4
Rounded-rounded	1	1	1
Fixed-rounded	2	1	1.2
Fixed-fixed	4	1	1.2

*To be used only with liberal factors of safety when the strip load is accurately known.

Source : Shigley (2006)

Two types of strip failure modes are well known. Local (flange) buckling dominates the behavior of stubby strip. Global (Euler) buckling controls the behavior of slender strip. For intermediate lengths, the local and global buckling modes interact leading to smaller failure loads than predicted by any of the two isolated modes acting alone (Thompson and Hunt, Ch. 12, 1973). In design, the interactive behavior for intermediate length strips is taken into account by an empirical constant (Barbero and Tomblin 1994). Prediction of such constant as a function of geometry, materials properties, and imperfections of the strip is the object of this investigation. For metallic strips the interaction takes place between one buckling mode and yield, but composites remain linear for large strains thus allowing for the interaction between different buckling modes.

Local buckling occurs in short strip a that are long enough not to fail due to crushing; that is, when the compression strength of the material is not reached (Barbero 1998a, Barbero 1999a, Barbero and Wen 2000). The short strip buckling load PL can be determined from a short strip test (Tomblin and Barbero 1994, Barbero and Trovillion 1998c) or predicted using analytical or numerical techniques (Bank and Yin 1996, Banks and Rhodes 1983, Barbero and Raftoyiannis 1990, Brown et al. 1998, Davalos and Qiao 1998, Pecce et al. 1998, Vakiener et al 1991, Yuan et al. 1991 etc.) Its value is independent of the length of the strip as long as the strip is short and wave amplitude modulation (Godoy 1999) is not a factor.

When the length of the strip is such that the predicted local and global loads are close, the experimental failure load is lower than both predictions, depending on the imperfections. This was shown experimentally by Barbero and Tomblin (1994) and Barbero et al. (1999b) for strip and by Foster (1981) for shells.

Iiyushin (1947) has treated the stability of plates. Stressed above the-elastic limit with consideration of three possible zones that might result from buckling as shown in Figure 2.3. A purely elastic zone, a zone n which part of the material is in the elastic and plastic state which called " elasto-plastic" zone and a purely plastic zone in which all of the plate is stressed beyond the elastic limit. All three zones may exist simultaneously if the plate is not entirely in the plastic state before buckling or if the buckling is allowed to proceed beyond the initial stages.

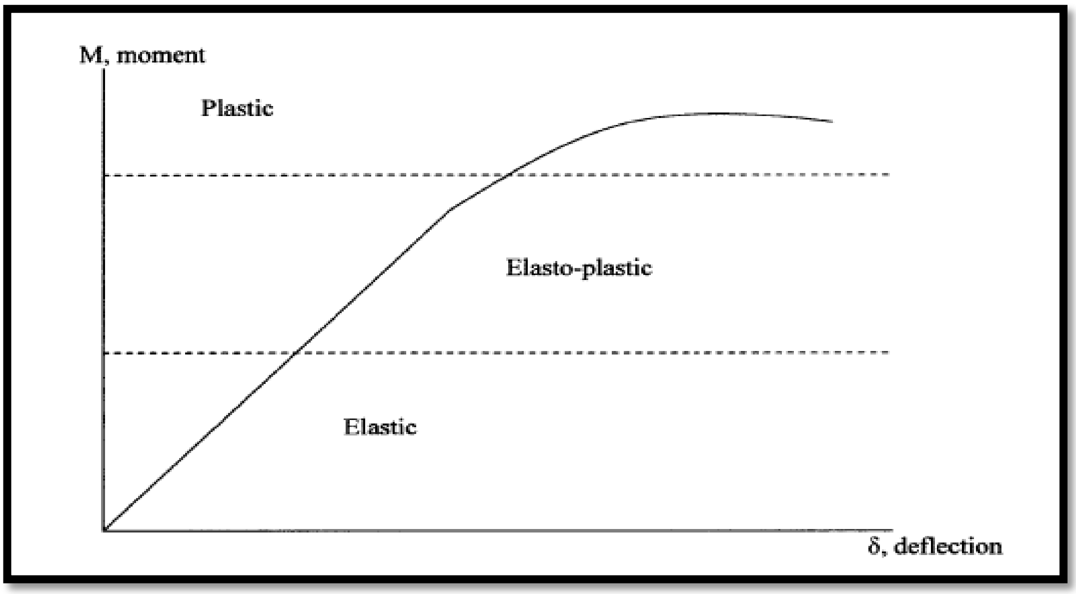


Figure 2.2: Graph Moment Versus Deflection

Source : Shigley (2006)

The elastic buckling and post-buckling behavior of plain channel section struts under compression and bending was examined by Rhodes and Harvey (1976) using a semi-energy method of analysis. This work outlines the essential differences in strut behavior when subjected to either controlled compression eccentricity or controlled load eccentricity. Centrically loaded struts are shown, in general, to have less stiffness after local buckling than uniformly compressed struts. Rhodes and Harvey (1976) point out that after local buckling, the uniformity of compression is lost for the case of centroidal loading and that due to the resulting changes in compression eccentricity, the post-buckling stiffness of the struts is reduced.

The investigation by Zbigniew Kolakowski (1998) is concerned with open cross-section strip under an axial compression or and a constant bending moment. The beam- strip are assumed to be simply supported at the ends. His aimed is to improve the study of the equilibrium path in the post-buckling behavior of imperfect structures with regarding the second order non-linear approximation and the effect of cross-sectional distortions. The principal goal of numerical analysis is to investigate the influence of the wall orthotropic factor of strips upon the all buckling modes from global (flexural, flexural-torsional, lateral, distortional and their combination) to local and upon the coupled post-buckling state. In the solution obtained the transformation of buckling modes with the increase of load up to the ultimate load and shear lag phenomenon is included.

2.4 Strip

A strip or pillar in architecture and structural engineering is a vertical structural element that transmits, through compression, the weight of the structure above to other structural element below. For the purpose of wind or earthquake engineering, strips may be designed to resist lateral forces.

A strut is a structural component designed to resist longitudinal compression. Struts provide outwards-facing support in their lengthwise direction, which can be used to keep two other components separate, performing the opposite function of a tie. They are commonly used in architecture and engineering. Ideal strip is a strip which does not show any deflection at the critical load for that strip i.e. P_{cr} is known as an ideal strip. In our real life o strips s is an ideal strip. A real strip is a strip which shoes some deflection even a minor one when critical load is applied.

When it comes to the classification of strips, there are 3 different standards on the basis of which we can classify and distinguish the strips. There are :

1. Based on L_e/D

   In this ratio, L_e represents the effective length of the strip and D represent the least dimension of the pillar.

2. Based on slenderness ratio

   Slenderness ration is the ratio between effective length and radius of gyration of the strip. Usually least radius of gyration is used for this equation as the strip buckle along the axis where there is least value of r and maximum value of slenderness ratio. This ratio can be mathematically expressed as :

$$\text{Slenderness ratio} = \frac{KL}{r}$$

3. Based on eccentricity.

   On the basis of this standard strip are to be divided in 2 categories :

   i. Strip with concentric loading
   ii. Strip with eccentric loading

The main parameter that will be investigate in buckling changes is a elongation buckle along the strip surface. By using FEM software, the strain in the elongation of strip can be investigate.

2.5 Stress Strain Diagram

A stress–strain diagram is a graph derived from measuring load (stress – σ) versus extension (strain-ε) for a sample of a material. The nature of the curve varies from material to material. The following diagrams illustrate the stress–strain behavior of typical materials in terms of the engineering stress and engineering strain where the stress and strain are calculated based on the original dimensions of the sample and not the instantaneous values. In each case the samples are loaded in tension although in many cases similar behavior is observed in compression.

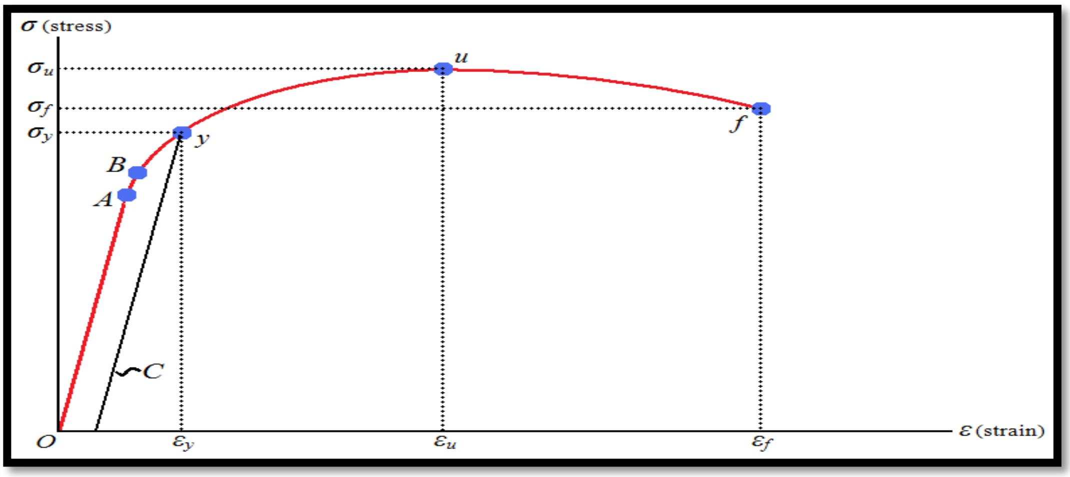


Figure 2.3: Stress Strain Diagram

(Sources : Shigley's (2006)

Stress-strain diagram obtained from the standard tensile test, where:

Point A marks the proportional limit;

Point B, the elastic limit;

Point y is the yield point;

Line ***C*** is determined from the offset method and is used to determine the yield point (point *y*);

The slope of line *OA* is equal to the modulus of elasticity;

Line *AB* is not a perfectly straight line, even though the specimen is elastic through this region

Stress-strain diagrams that use data from tensile tests are engineering stress-strain diagrams because the stresses and strains calculated from the data are not true values. The stress is based on the original area prior to the load being applied. In reality, as the load is applied the area reduces so that the actual or true stress is larger than the engineering stress. To obtain the true stress for the diagram, the load and the cross-sectional area must be measured concurrently during the test. Typically, the true stress is much higher than the engineering stress at the necked section.

2.6 Finite Element Method

Finite element method (FEM) is a numerical procedure that can be used to obtain solutions to a large class engineering problems involving stress analysis, heat transfer, vibration, deflection, buckling behavior, and many other phenomena. Besides that, the Finite element method, it is also a numerical technique for finding approximate solutions of partial differential equation as well as of integral equations. The finite element formulation of the problem results in a system of simultaneous algebraic equations for solution, rather than requiring the solution of differential equations. These numerical methods yield approximate values of the unknowns at discrete numbers of points in the continuum. Hence this process of modeling a body by dividing it into an equivalent system of smaller bodies or units (finite elements) interconnected at points common to two or more elements (nodal points or nodes) and/or boundary lines and/or surfaces is called discretization. In the finite element method, instead of solving the problem for the entire body in one operation, it can formulate the equations for each finite element and combine them to obtain the solution of the whole body. Which are then numerically integrated using standard techniques such as Euler's method, and Runge-Kutta.

Finite element analysis (FEA) has become commonplace in recent years, and is now the basis of a multibillion dollar per year industry (Brebbia, 1982). Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even introductory treatments of Mechanics of Materials. In spite of the great power of FEA, the disadvantages of computer solutions must be kept in mind when using this and similar methods. There is a lot of software available for Finite Element Analysis, such as Algor (FEMPRO), ANSYS, ABAQUS, and MSC Patran. All this software widely used nowadays, in order to analyze the part. In solving partial differential equations, the primary challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors in the input and intermediate calculations do not accumulate and cause the resulting output to be meaningless. There are many ways of doing this, all with advantages and disadvantages. The Finite Element Method is a good choice for solving partial differential equations over complicated domains.

Among other methods, the FEM is widely used in metal forming analysis due to its capabilities to model the complicated shapes of tools and parts in forming processes. Hence we choose the finite element software ANSYS because based on ANSYS User's Manual for Rev. (1995), it state that the finite element software ANSYS is the commercial used.

FEM is a numerical procedure that can be used to obtain solutions to a large class engineering problems involving stress analysis, heat transfer, vibration, deflection, buckling behavior, and many other phenomena (Osman, 2004; Marcus *et al.*, n.d.; Lira *et al*., 2002). This method can be used to analyze either small or large-scale deflection under loading or applied displacement. The simulation is required because of the astronomical number of calculations needed to analyze a large structure. The finite element analysis is a way to deal with structures which are more complex than can be dealt with analytical solution using classical theories (Chan *et al*., 2006; Hertzberg, 1996). The key idea of the finite element method is to discretize the solution domain into a number of simpler domains called elements. An approximate solution is assumed over an element in terms of solution at selected points called nodes (Bhatti, 2005).

Stephen and Steven [6-9] derived an error measure for the buckling and natural frequency finite element analysis. This was based on the more accurate eigenvector solution to determine an error measure for the eigenvalue representing the buckling load or natural frequency of a structure. The previous papers [6-9] only examined the beam elements and two-dimensional in plane plate elements. These elements have simple shape functions which lead to relatively high errors.

For this project, by using FEM, the critical region, and strain force will be the parameter that will be investigate. The data of parameters will be collected and analysis by using software ABAQUS version 6.7, At the end, the result from experimental test equipment will be compare to the model simulation.

2.7 Fabrication

A fabricator's work starts from the point of procurement of raw materials including fasteners and ends with the dispatch of the fabricated items to site for erection. In order to ensure that the fabrication can be carried out in accordance with the drawings, it is necessary that inspection and checking is carried out in accordance with an agreed Quality Assurance Plan (QAP). The plan should elaborate on checks and inspections of the raw materials and also of the components as they are fabricated, joined etc. During the last two decades, fabrication activities have increased steadily in yards adjacent to work. In the absence of controlled environment (as in an organized workshop), the quality of workmanship of such fabrication is likely to suffer. It has, therefore, become all the more important to motivate the fabricators to appreciate the usefulness of Quality Assurance Plans and introduce the system in all their works and at site as well.

2.7.1 Imperfections in Fabrication

Structural steelwork cannot be fabricated to exact dimensions and some degree of imperfection is bound to occur during fabrication process. The limits of various imperfections are spelt out in the specifications. In the design, these are accounted by adopting a factor of safety for material. However, in some components an increase of imperfection beyond these limits may lead to reduction in the strength and durability of the structure e.g., imperfections on the straightness of the individual flanges of a rolled beam or a fabricated girder results in the reduction of strength of the girder due to lateral torsional buckling which may cause an overall bow in the girder. This, in turn, may generate twisting moments at the supports. As a rule all strips and struts should be checked for straightness on completion of fabrication. Also, all rolled and fabricated girders should be checked over a distance in the longitudinal direction equal to the depth of the section in the region and points of concentrated load.

2.8 Welding

Welding is a fabrication process that joins materials, usually metals or thermoplastics, by causing coalescence. This is often done by melting the workpieces and adding a filler material to form a pool of molten material (the weld puddle) that cools to become a strong joint, with pressure sometimes used in conjunction with heat, or by itself, to produce the weld. This is in contrast with soldering and brazing, which involve melting a lower-melting-point material between the workpieces to form a bond between them, without melting the workpieces. Many different energy sources can be used for welding, including a gas flame, an electric arc, a laser, an electron beam, friction, and ultrasound. While often an industrial process, welding can be done in many different environments, including open air, underwater and in outer space. Regardless of location, however, welding remains dangerous, and precautions must be taken to avoid burns, electric shock, eye damage, poisonous fumes, and overexposure to ultraviolet light.

A welded joint is made by fusing (melting) the steel plates or sections along the line of joint. The metal melted from each member of the joint unites in a pool of molten metal, which bridges the interface. As the pool cools, molten metal at the fusion boundary solidifies, forming a solid bond with the parent metal. When solidification completes, there is a continuity of metal through the joint. There are five welding process regularly employed namely:

a) Shielded Metal Arc Welding (SMAW)
b) Submerged-Arc Welding (SAW)
c) Manual Metal-Arc welding (MMA)
d) Metal-Active Gas welding (MAG)

2.8.1 Methods of welding

In this project, we will use Shielded Metal Arc Welding (SMAW). This is basically a semi-automated or fully automated welding procedure. The type of_welding electrode used would decide the weld properties. Since this welding is carried_out under controlled condition, the weld quality is normally good.

2.8.2 Defects in welds

Faulty welding procedure can lead to defects in the welds, thereby reducing the strength of the weld. Fig.*8* shows some of the common defects in welds. Some of these are:

i. Undercut
ii. Porosity
iii. Incomplete Penetration
iv. Lack of side wall fusion

It should be emphasized that a 'theoretical 100% error free' weld is not achievable in practice. While good quality welds are the priority of welders and weld inspectors minor defects do normally creep in. Hence these defects are assessed during a welding inspection. If the defects are within acceptable limits, they are accepted. If not, alternative measures of rectification may have to be carried out. Table *2* shows nature of some of the defects and their acceptability limits.

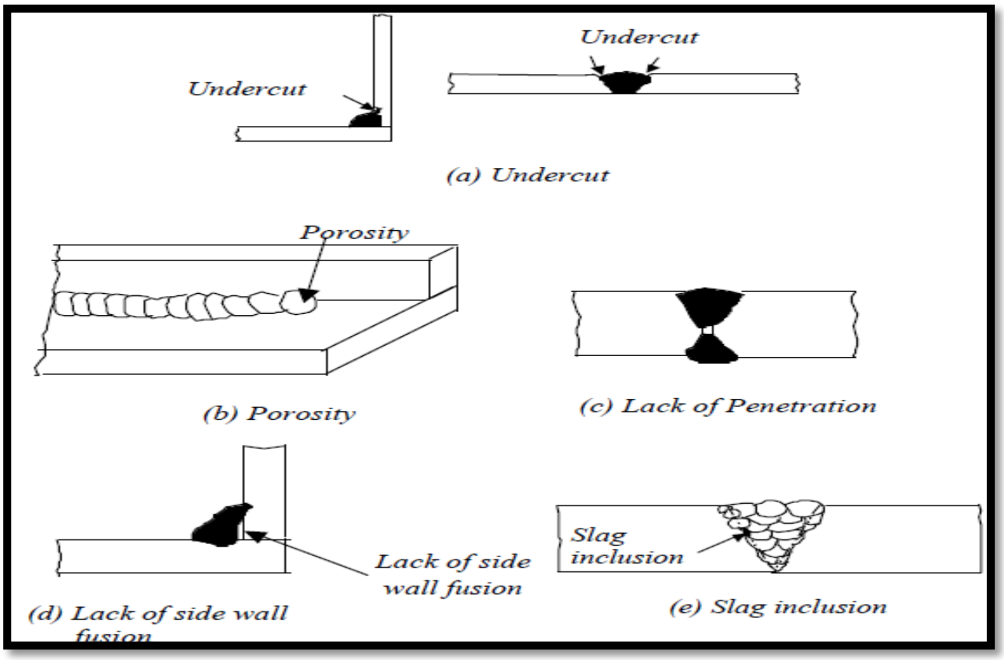


Figure 2.4: Defects In Welds

(Source : Pratt.J.L., " Introduction to the Welding of Structural Steelwork", 3rd revised edition, The Steel Construction Institute, 1989.)

2.9 The Strain Gauge

The metallic strain gauge consists of a very fine wire or, more commonly, metallic foil arranged in a grid pattern. The grid pattern maximizes the amount of metallic wire or foil subject to strain in the parallel direction (Figure 2). The cross sectional area of the grid is minimized to reduce the effect of shear strain and Poisson Strain. The grid is bonded to a thin backing, called the carrier, which is attached directly to the test specimen. Therefore, the strain experienced by the test specimen is transferred directly to the strain gauge, which responds with a linear change in electrical resistance. Strain gauges are available commercially with nominal resistance values from 30 to 3000 Ω , with 120, 350, and 1000 Ω being the most common values.

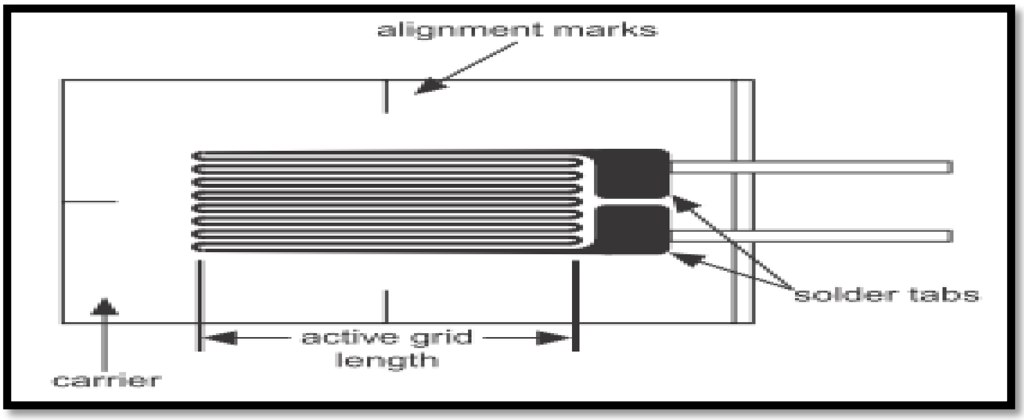


Figure 2.5: Strain Gauge

(Source : Strain Gauge Measurement - A Tutorial)

It is very important that the strain gauge be properly mounted onto the test specimen so that the strain is accurately transferred from the test specimen, though the adhesive and strain gauge backing, to the foil itself. Manufacturers of strain gauges are the best source of information on proper mounting of strain gauges. A fundamental parameter of the strain gauge is its sensitivity to strain, expressed quantitatively as the gauge factor (GF). Gauge factor is defined as the ratio of fractional change in electrical resistance to the fractional change in length (strain):

$$\boxed{GF = \frac{\Delta R/R}{\Delta L/L} = \frac{\Delta R/R}{\epsilon}}$$

2.8 Summary of literature review

This sub-topic is a points of literature review that I choose for project. This points that have been choose is related to teaching aid, buckling and FEM.

Author	Title	Research point
Dr. Burl E. Dishongh	"Paradigm for Teaching Structural Technology". Journal of Construction Education, Spring 2003, Vol. 8, No.1, pp 28-37	Another research has studied two paradigms for teaching structures and suggested a paradigm that focuses on appreciation of engineering processes and use of sketches, experimental observations, and basic mathematic and physical relationship.
R. Kopp, M.L. Cao and M.M. Souza,	"Muti-level simulation of metal forming process", Proc. 2nd ITCP, Stuttgart, August 1987, pp. 1128-1234.	Computer simulation has become reliable and acceptable in the metal forming industry since the 1980's. Metal forming analysis can be performed in three modeling scales .
Shigley.	"Shigley's Mechanical Engineering Design", 8th edition,2006, pp. 177-178.	The factor C is called the end-condition constant, and it may have any one of the theoretical values 1/4, 1, 2, and 4, depending upon the manner in which the load is applied. Even if the ends are welded, some deflection will occur. When axial load applies at the strip, the critical load for a slender strip can be determined by the Euler

		Buckling Load.
Shigley.	"Shigley's Mechanical Engineering Design", 8th edition,2006, pp. 144.	The problem of bending of beams probably occurs more often than any other loading problem in mechanical design. Shafts, axles, cranks, and wheels, as well as many other elements, must often be treated as beams in the design and analysis of mechanical structures and systems.
Shigley.	"Shigley's Mechanical Engineering Design", 8th edition,2006, pp. 174.	The critical unit load. It is the load per unit area necessary to place the strip in a condition of unstable equilibrium. In this state any small crookedness of the member, or slight movement of the support or load, will cause the strip to begin to collapse.
Foudil Mohri, Cherif Bouzerira, Michel Potier-Ferry.	"Lateral buckling of thin-walled beam- strip elements under combined axial and bending loads", 20 October 2006;	The proposed analytical solutions have been compared to linear stability ones and to non-linear finite element results using shell elements. Numerical buckling moments have been computed from singular points observed along the non-linear equilibrium paths. These solutions are close to FEM results.
Osman	2004; Marcus *et al.*, n.d.; Lira *et al.*	The Finite Element Method (FEM) is a numerical procedure that can be used to obtain solutions to a large class engineering problems involving stress analysis, heat transfer, vibration, deflection, buckling behavior, and many other phenomena
Sérgio Tonini Button	"Numerical Simulation and Physical Modeling as Educational Tools to Teach Metal Forming Processes"	The Finite Element Methods (FEM) is widely used in metal forming analysis due to its capabilities to model the complicated shapes of tools and parts in forming processes. Besides that the finite element software ANSYS is the commercial used.
David Roylance	"Finite Element Analysis", February 28,	Finite element analysis (FEA) has become common place in recent years, and is now the basis of a

	2001	multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even introductory treatments of Mechanics of Materials.
D. B. Stephen, G.P. Steven	Error estimation for plate vibration elements, Research report, FEARC-9408, Finite Element Analysis Research Centre, University of Sydney (1994).	They derived an error measure for the buckling and natural frequency finite element analysis. This was based on the more accurate eigenvector solution to determine an error measure for the eigenvalue representing the buckling load or natural frequency of a structure.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this chapter 3, the method to conduct this thesis is being described in detail. Besides that, this chapter includes the flow of the research paper, information on preparation of lateral bending test equipment, the flow of experiment done using the suitable apparatus, and method of interpreting the result. The experiment is suggested to design and fabricate of a lateral bending test equipment with both ends are pinned. This Figure 3.1 shows overall research activities flow chart.

There are brainstorming was conducted by the researcher and supervisor for the design of lateral bending test equipment and how to write a report. For examples, how to find a journal, what format used for report and what kind of design that better and it was conducted during week 3 until week 8. The journal resource that used is came from Science Direct, Scopus, Springer Link and ELT Journal. Methodology flow chart is use as guidelines and the sequences to make this project go smoothly. Before that, the process begin with title selection and conformation of the title selected. Then, the process continue with finding the articles relate and identify the objectives, project background and the problem statement related to this project. The process proceeds by starts with design/sketch and identifies the material needed to fabricate the test equipment for this lateral bending experiment.

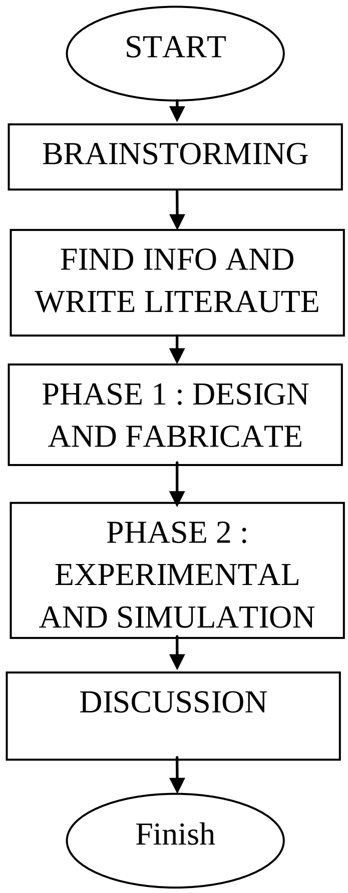


Figure 3.1: Overall Activities Flow Chart

In the phase 1, the researcher is focused on design and fabricate the buckling test equipment which is conducted in semester 1 and semester 2. It is include the sketching, material selection, machining, joining, welding, finishing and improving the model.

For phase 2, which is conducted in semester 2, the researcher is focused on design by using SOLIDWORKS and ABAQUS simulation, then proceed with testing the model and specimens to compare the buckling behavior with the simulation model. After that, the gauge strain were used to get the value of strain on critical region of the specimens. Then, data logger are used to convert the electric displacement from the strain gauge by using software DASYLab 10.0 and Measurement & Automation to get the data of strain in graph.

3.2 Flow Chart

Phase 1 : Design and fabricate

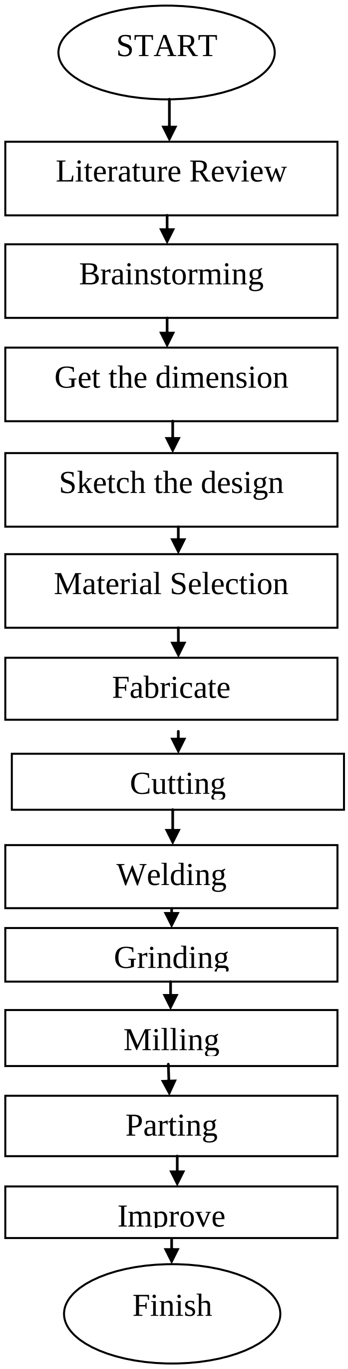


Figure 3.2: Design and Fabricate Flow Chart

Phase 2 : Experimental and simulation

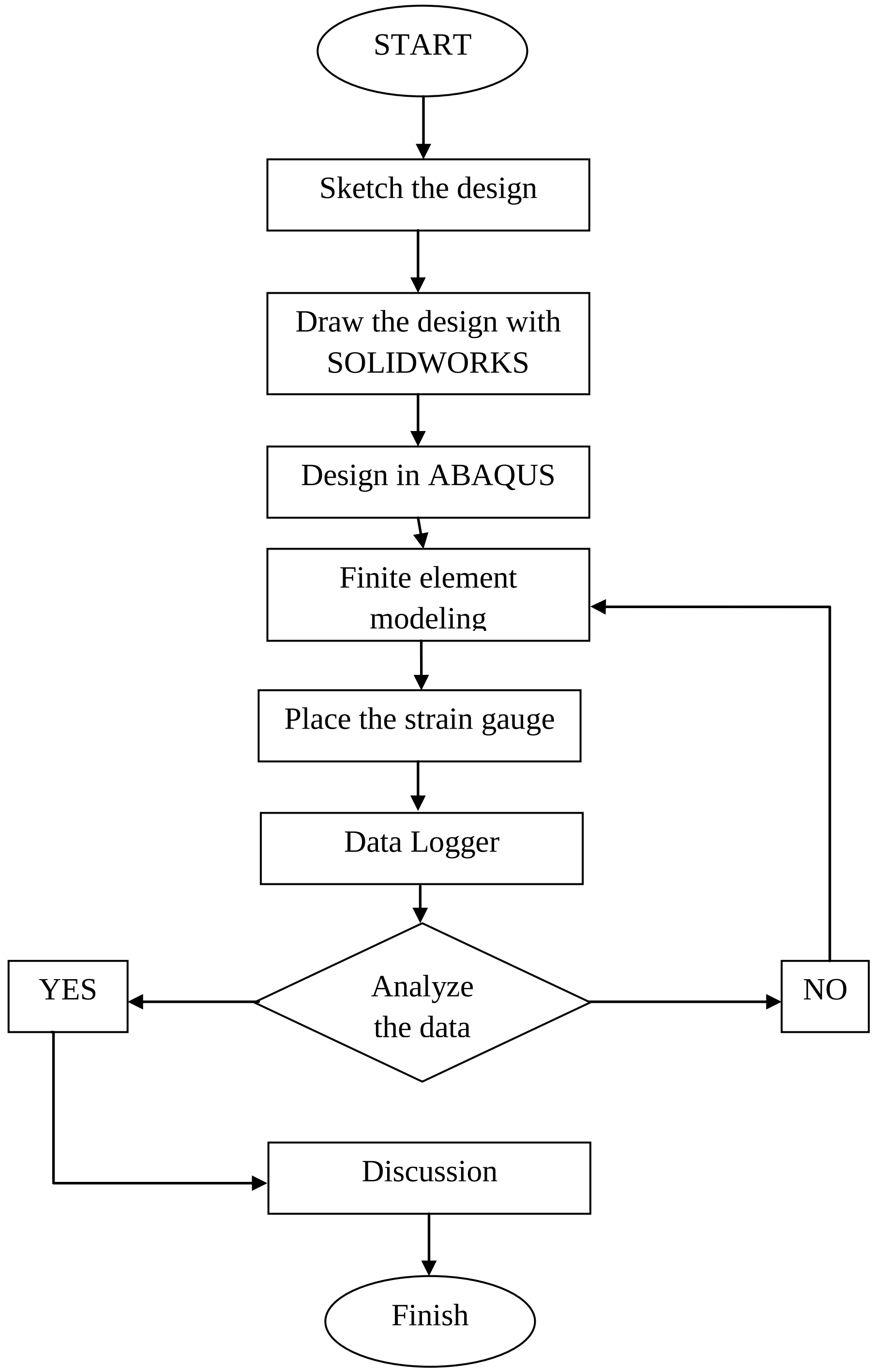


Figure 3.3: Experimental and Simulation Flow Chart

3.3 Design Equipment

SOLIDWORKS is a Parasolid-based solid modeler, and utilizes a parametric feature-based approach to create models and assemblies. Building a model in SOLIDWORKS usually starts with a 2D sketch (although 3D sketches are available for power users). The sketch consists of geometry such as points, lines, arcs, conics (except the hyperbola), and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parametric nature of SOLIDWORKS means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside of the sketch. Thus, the selection material had been decide in which for the basic base, the material that been used is rectangular hollow steel (50mm x 50mm x 250mm),the upper base is also rectangular hollow steel (38mm x 38mm x 250mm).For the side support, the material used is L-shape steel (25mm x 25mm x 600mm).The additional material for the structure are solid aluminum (50mm x 50m x 150mm),sheet metal (250mm x 600mm x 3mm) for the cover of the main structure. Lastly for the specimen of buckling test, the material used is 0.5mm Galvanize Iron (0.5 GI). The design of material selection can be refer to the appendix.

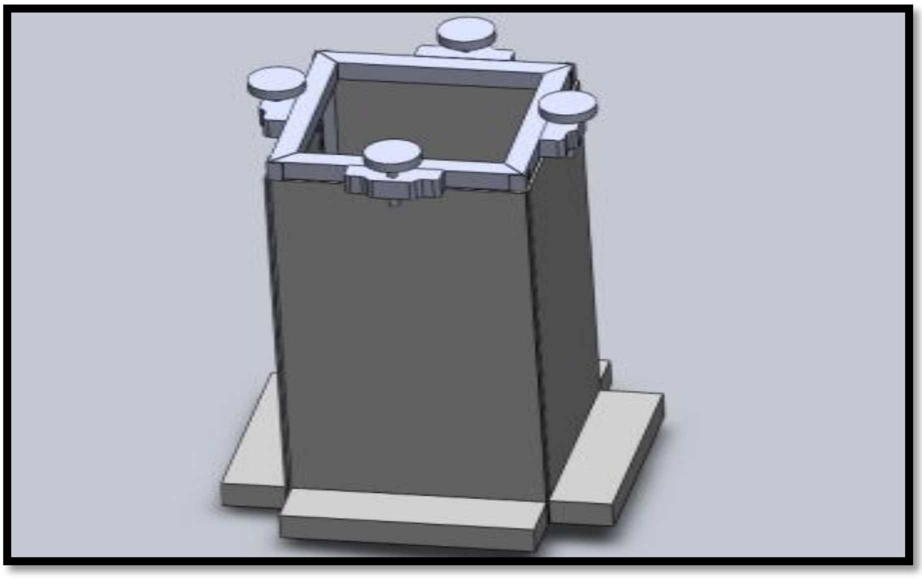


Figure 3.4: Design Buckling Test Rig

3.4 Fabrication Process

After the designing phase, here comes the fabrication process. This process is about using the material selection and makes the product base on the design and by following the design dimension stated in the Bill of Material (BOM). Many methods will be used to fabricate the product such as measuring and marking, cutting, welding, drilling, joining and finishing. Fabrication process is a process to make one product rather than manufacturing process that focus to large scale production. As there are a lot of processes of fabrication, there also need a lot of machines and tools to perform the processes.

Table 3.1: Bill of Material

No	Parts	Dimension (mm)	Quantity
1	Rectangular Hollow (Steel)	50 x 50 x 250	4
2	Rectangular Hollow (Steel)	50 x 50 x 250	4
3	L-shape steel	25 x 25 x 600	4
4	GI Sheet metal	250 x 250 x 3	4
5	Solid Aluminum	35 x 35 x 80	1
6	Solid Aluminum	50 x 50 x 80	1
7	Solid Aluminum	D=15 L=100	1
8	Aluminum	D=80 t=10	1
9	Rivet	5	6
10	Screw	5 x 60	2

3.5 Machine and Process of Fabricate.

In this project, there are a lot of machine, tools and process that required to fabricate the product. Under this sub-topic, it will show the pictures and a little bit explanation of the machine that involved in this project.

3.5.1 Band Saw

This machine was capable to cut solid materials. In this project, we used it to cut solid round and rectangular aluminum bar. This machine can be operate manually and automatic by only switching the button.

Figure 3.5: Band Saw

3.5.2 Vernier Height Gauge and Angle L Square Ruler

Before we cut the material to the dimension we want, we need to measure and mark the material first. This is to ensure precision of the material's length which is quite important in the fabrication process. it is to avoid the waste of the material because w undersize while cutting it. To perform this process, we will need vernier height and angle L square ruler.

Figure 3.6: Measuring Tools

3.5.3 Disc Cutter

This disc cutter was used to cut the materials according to the measuring that have been marked. It can be used to cut rectangular hollow L bar steel. This machine can used to cut with angle of 0 and 45 degree.

Figure 3.7: Disc Cutter

3.5.4 Shield Metal Arc Weld (SMAW)

In this project, this welding machine was used to join the rectangular steel bar to be a steel frame. This is basically a semi-automated or fully automated welding procedure. The type of welding electrode used would decide the weld properties. Since this welding is carried out under controlled condition, the weld quality is normally good.

Figure 3.8: Shield Metal Arc Weld (SMAW)

3.5.5 Shearing Machine

This machine was used to cut the sheet thin wall with the dimension 250 x 250 mm. The shearing machine were used hydraulic power to press the cutter.

Figure 3.9: Shearing Machine

3.5.6 Milling Machine

This project involve in milling process which is to shape the rectangular aluminum bar to the upper holder that will attach to the upper base. Thus, in this process, the milling machine will be used. This machine also can do a surface finishing for the product in flat surface. The material can be turned to various orientations in the middle of the process and cutting tools of various shapes can be used.

Figure 3.10: Milling Machine

3.5.7 Turning Machine

Turning is a form of machining, a material removal process, which is used to create rotational parts by cutting away unwanted material. The turning process requires a turning machine or lathe, work piece, fixture, and cutting tool. In this project, this machine is need to shaped the aluminum solid bar into shaft aluminum.

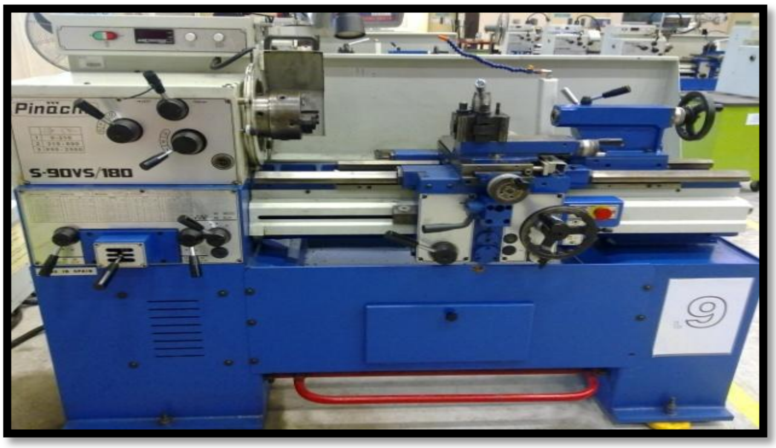


Figure 3.11: Turning Machine

3.5.8 CNC Machine

This machine was very useful for make a complicated design. In this project, this machine was use to shape aluminum bar into upper holder aluminum. To shape it, we just need to do design in MasterCam then transfer to the computer for the shape details.

Figure 3.12: CNC Machine

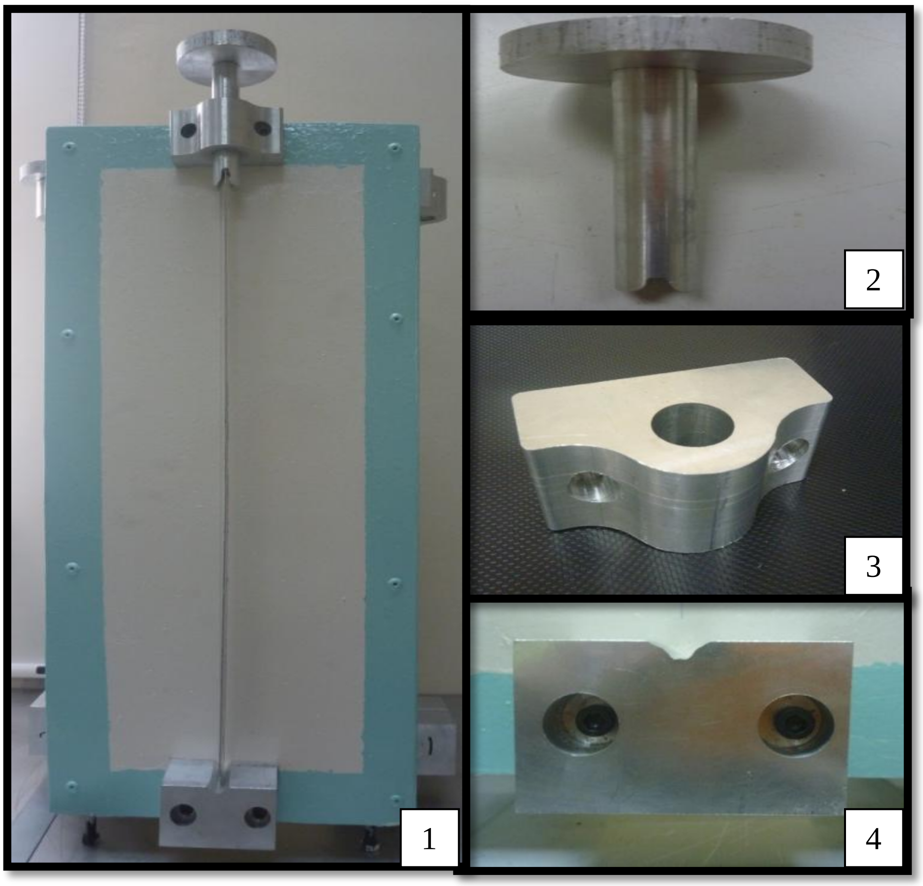
3.5.9 Drilling Machine (Press Drill)

For this machine, materials need to be drill at several locations to make a hole for rivet, bolts and nut. It is more easy rather than hand drill because it can give more force to the material.

Figure 3.13: Press Drill

3.6 New Test Rig of Buckling Test (both end condition are pinned)

By follow early design and fabrication process, machining and finishing, the test rig of buckling test had been produce.



Notation:

[1] Product (Buckling Test Rig).

[2] Load holder with end design.

[3] Upper holder.

[4] Lower end design.

Figure 3.14: Final Product

3.7 Experimental of Lateral Bending (Buckling)

The experiment of the lateral bending (buckling) test will be run after all setup of the structure equipment had been done. In this experiment, the specimen that will be use is 0.5 GI (0.5 Galvanize Iron) with length of 700mm.The experiment purpose is to get the bending value or in other words is the value of the strain when the buckling phenomenon occur. First, the load will be added on the top of the load holder and the force will directly to the specimen. The specimen is located in condition of both ends are pinned. The changes of the straight shape specimen to the buckle shape will be capture and the test read of the experiment will be record. The value of the strain can be measure by using strain gauge that placed on the critical region of specimen. Finally, the result from strain gauge will be compare with FEM by using software ABAQUS.

3.7.1 Boundary Condition

The experiment is to fabricate buckling test rig under both end are pinned condition. For pinned condition, the design at end condition is to make a "V" shape because at that condition, the specimen will have zero displacement at y, x and z axis but have a moment value at y, x and z.

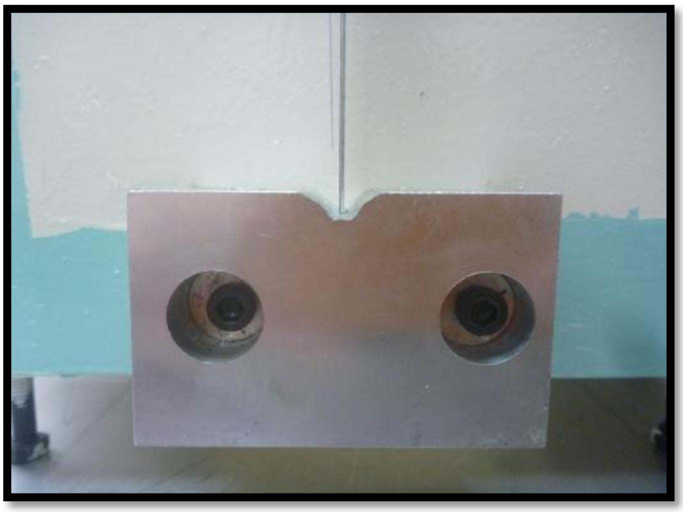


Figure 3.15: Bottom Pinned Condition

Figure 3.16: Upper Pinned Condition

3.7.2 Strain Gauge

Strain (ε) is the amount of deformation of a body due to an applied force. Strain can be positive (tensile) or negative (compressive). Although dimensionless, strain is sometimes expressed in units such as in./in. or mm/mm. In practice, the magnitude of measured strain is very small. Therefore, strain is often expressed as microstrain (με). Specifically, strain is the fractional change in length, as shown in the following figure below.

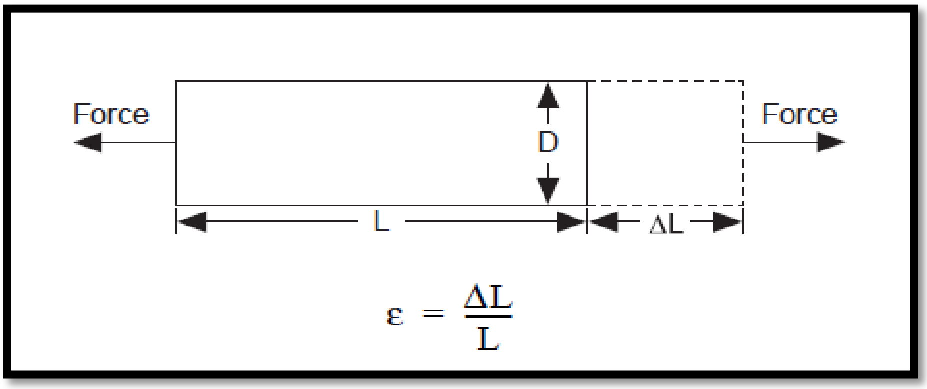


Figure 3.17: Strain Gauge

The strain gauge was placed at critical region when buckle happen. The installation of strain gauge are required following items for bonding and lead wire connection: Strain gauges, bonding adhesive, connecting terminals, test specimen, solvent, cleaning tissue for industrial use, soldering iron, solder, abrasive paper (120 - 320 grit), marking pencil, scale, tweezers, extension lead wire, polyethylene sheet, nippers. To positioning, roughly determine the location on the test specimen where the strain gauge is to be bonded, it is on critical region. Before bonding, remove all grease, rust, paint, etc., from the bonding area. Sand an area somewhat larger than the bonding area uniformly and finely with abrasive paper. Finish the surface with #120 to 180 abrasive paper for steel, or #240 to 320 for aluminum.

After that, do a fine cleaning and then apply bonding adhesive at the strain gauge. Then place the gauge on the guide mark, place a polyethylene sheet onto it and press down on the gauge constantly using your thumb or a gauge pressing device. This should be done quickly as the curing process is completed very fast. The curing time varies depending on the gauge, test specimen, temperature, humidity and pressing force. The curing time under normal conditions is 20 - 60 seconds. After that, wrap the gauge leads around the connecting terminal wires.

Solder the junction area with a little slack in the gauge leads, taking care to prevent excessive tension during measurement. Finally, solder an extension lead wire to the terminal wires on the opposite side of the connecting terminals. Then clip off any excess extension lead wire with a pair of pliers or wire cutters.

Figure 3.18: Installation Strain Gauge

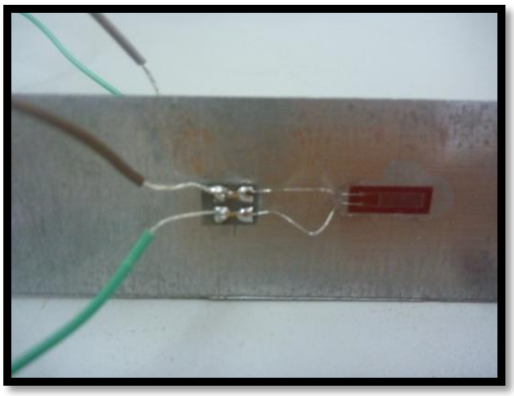


Figure 3.19: Finish Installation Strain Gauge

3.7.3 Data Logger

Data logger are available for a variety of sensor measurements including thermocouples, RTDs, strain gages, load and pressure transducers, torque cells, accelerometers, flow meters, and microphones. NI Compact DAQ systems combine sensor measurements with voltage, current, and digital signals to create custom, mixed-measurement systems with a single, simple USB cable back to the PC, laptop, or notebook. In this experiment, we use data logger NI-DAQ-9178 and use it function to sensor the strain gauge at the specimen.

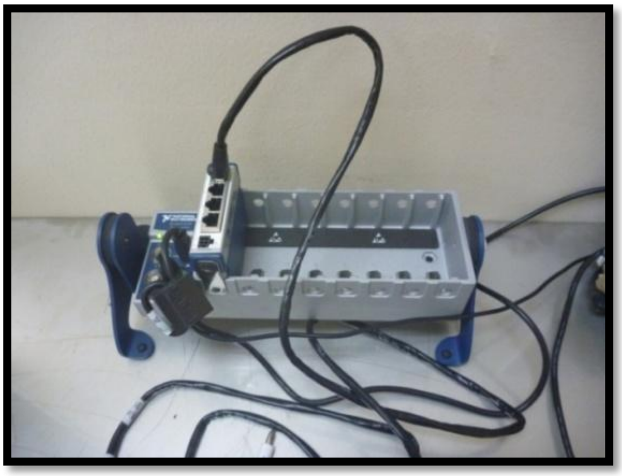


Figure 3.20: Data Logger

Figure 3.21: Screw Terminal Accessory

3.7.4 DASYLab Software

DASYLab is a software to read data from data logger. The value of strain that get from specimen buckle will be read by strain gauge sensor, then strain gauge will transfer the data to the data logger. Finally, DASYLab will convert the data into graph and value. To run the experiment, there are need to install software measurement and automation and DASYLab in laptop. First, open the software measurement and automation to check the whether the strain gauge is functional or not. Then, open the DASYLab software and do the diagram as Figure 3.24 to get the output from data logger. The following flowchart depicts the main steps required in an NI-DAQmx application to measure strain. Alternatively, we can configure a task to measure strain with a strain gage using the DAQ Assistant.

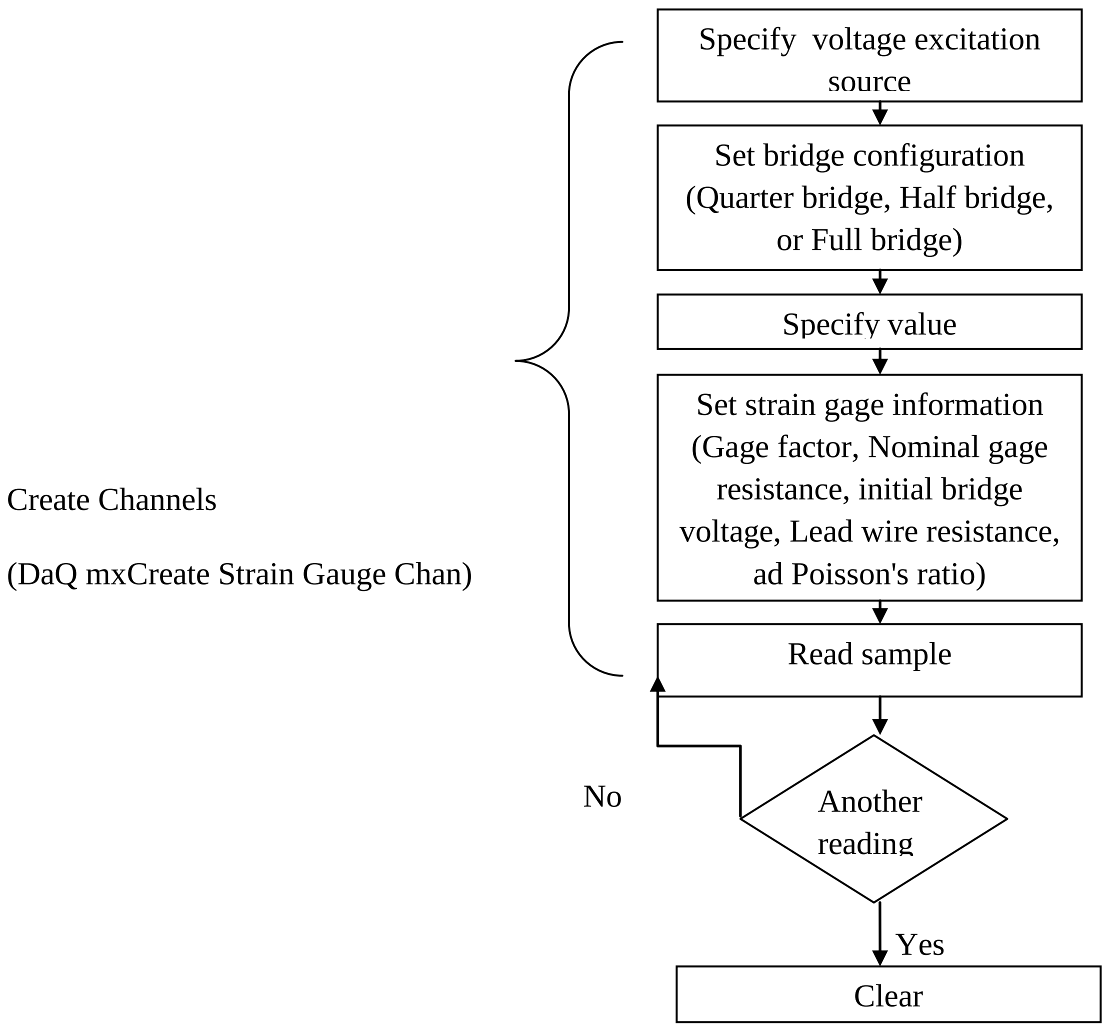


Figure 3.22: Measuring Strain Flow Chart

The figure below show the diagram of connection strain gauge in DASYLab software. The computer-based instrument acquires data in a rather straightforward system. A sensor, for example a strain gauge, picks up the “event” or real world property, the strain. This analog information is received by the data acquisition hardware and is digitized; this is known as “Analog to Digital conversion”. The digitized value is sent to the computer and received by the data acquisition software. The software then interprets the value, conditions, scales, displays and stores the data. The figure 3.25 show the diagram that need to do when run the DASYLab software. The figure 3.26 is diagram to check whether the strain gauge is functional.

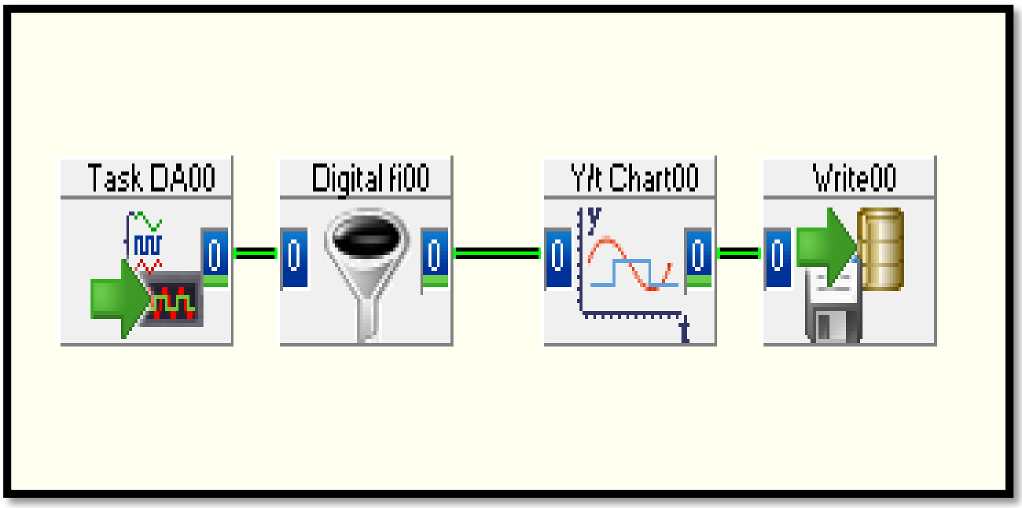


Figure 3.23: Diagram of Connection Strain Gauge in DASYLab Software.

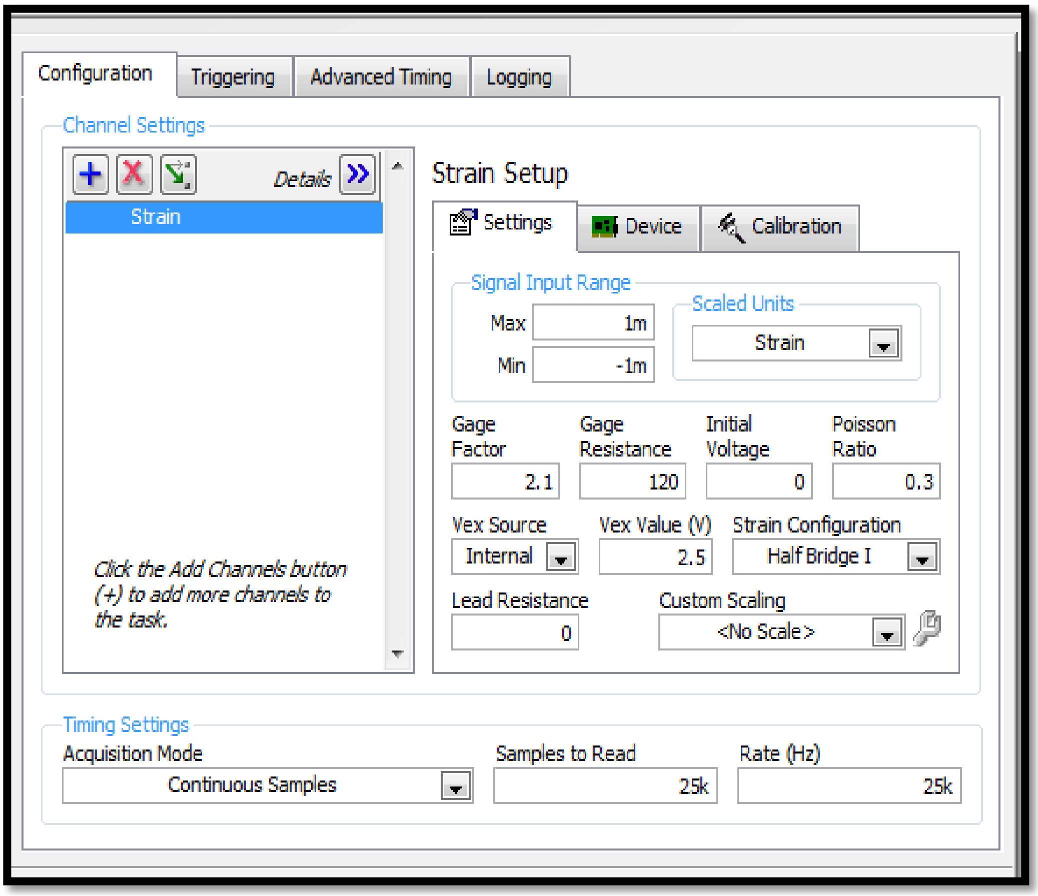
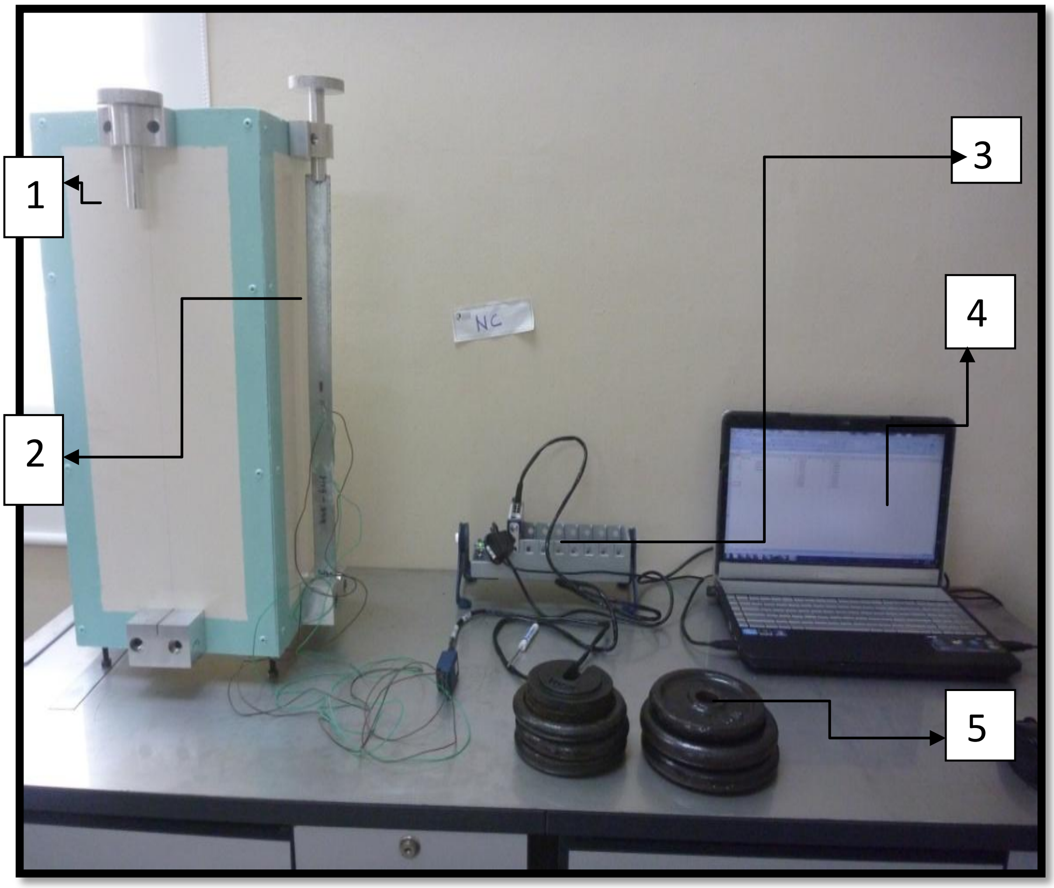


Figure 3.24: Diagram of Connection Strain Gauge in Measurement and Automation Software

3.8 Test Preparation



Notation:

[1] Product (Buckling Test Rig)

[2] Specimen

[3] Data Logger

[4] Laptop with software DASYLab & Measurement And Automation

[5] Load

Figure 2.25: Test Experiment Apparatus

From the figure above, it show the completed setup to run the experiment. The load have 0 N until 20 N and the laptop must have software DASYLab and Measurement And Automation to run the experiment.

3.9 Test planning

For this experiment, we would like to compare the physical behavior of buckle between experimental and simulation modeling, it is to verify whether the product can be use as buckling test rig. The simulation modeling is built up by using software ABAQUS 6.7, then the physical behavior of buckle will be recorded during the experiment in varies load.

The experiment had been plan to run by using starting load, 1 N until 20 N for this buckling test in which both ends are pinned. The experiment of the each load will be run for three times and there are will a pre-test experiment. It is to identify the critical region at the specimen surface, for strain gauge location. We use strain gauge as a device to get strain value from the specimen critical region when buckle.

The data of plain strain at the surface and maximum displacement at the critical buckle point gain will be collect. The value of each experiment will be total up to take the average value of each load given, it is because to reduce the error during the experiment. After the data completed filled, we will compare the data with the simulation data in ABAQUS.

Table 3.2:Test Planning

Load (N)	**Test 1**	**Test 2**	**Test 3**	**Average**
0				
5				
10				
15				
20				

3.10 Finite Element Analysis

3.10.1 ABAQUS Software

By using ABAQUS, there are few assumption that need to list. First, the model experimental is assume to initially perfectly straight, the load is applied through the center of the cross section, the material is homogeneous and linear static and the model is buckle under perfect condition and bend in a single plane. The modeling should have end pinned condition to validate to experimental model.

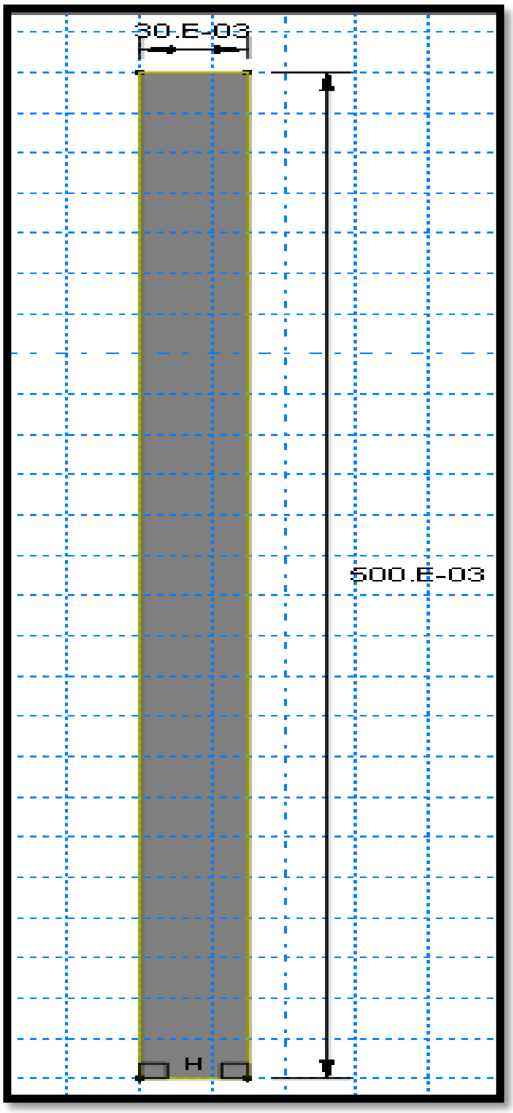


Figure 3.26: Simulation Model in ABAQUS

ABAQUS models are defined in terms of geometric features that must be subdivided into finite elements for solution. This process of sub division is called meshing. Mesh datasets contain information about element types, element discretisation and mesh type which is C4PE, Continuum 4 Plane Strain are used. The analysis type is buckling test and its step is single step. which is static. The condition is U1=U2=U3=0.

3.10.2 Modeling

The specimen models were assigned galvanized iron steel for its material property with Young's modulus, E= 2e11 N/mm2 , Yield Strength, is 203943242.6 N/mm2, Density is 7870 kg/m^3 and Poisson ratio of 0.29. The material properties resource is get from SOLIDWORKS 2011, the table can refer to Appendix B2. Load is given to the model by using single step properties because of static condition of simulation. By using buckling type of the analysis, the model buckle depend on load apply at the top of the model in ABAQUS.

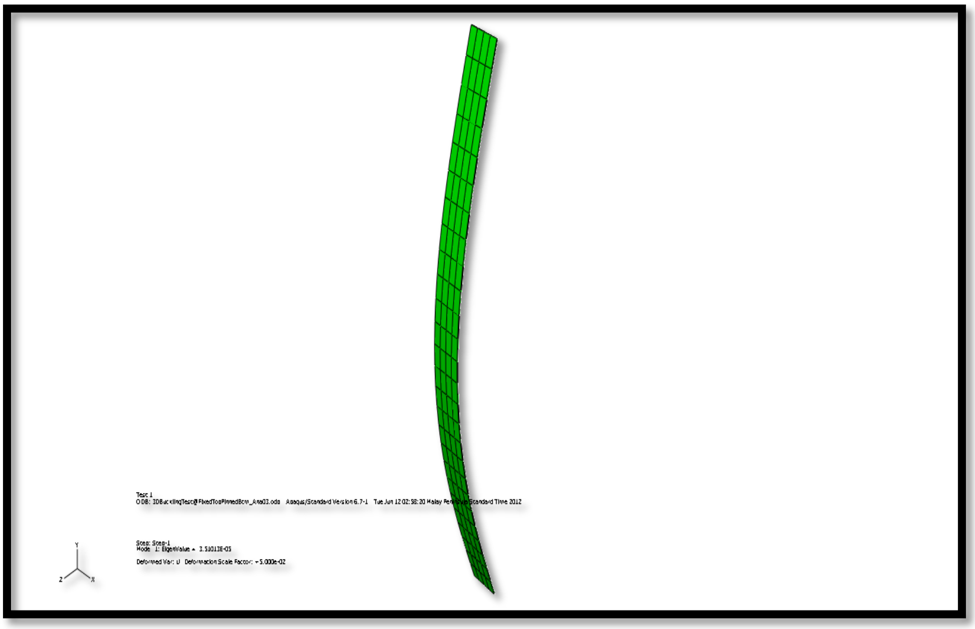


Figure 3.27: Simulation Model Under Buckle

3.11 Chapter Summary

In this chapter 3, there are been discuss about the flow of fabrication process for the buckling test equipment and the machine used for the fabrication process. There are also discussing about the flow of the project which is starting with come out new design and selection of material. Then, the result from the experimental will be compare with the simulation by using FEA software as the verification for the experimental result.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

In this chapter 4 presents the new buckling test rig which both end condition are pinned that have been fabricate. This product will be tested to validate whether it can be used as buckling test rig. The data from experimental and simulation will be collected to be compared. The experimental data that were collect are plain strain value and maximum displacement at the critical point of specimen's buckle. The simulation will demonstrate the critical region, strain value and physical behavior for every additional load.

4.2 Experimental Result

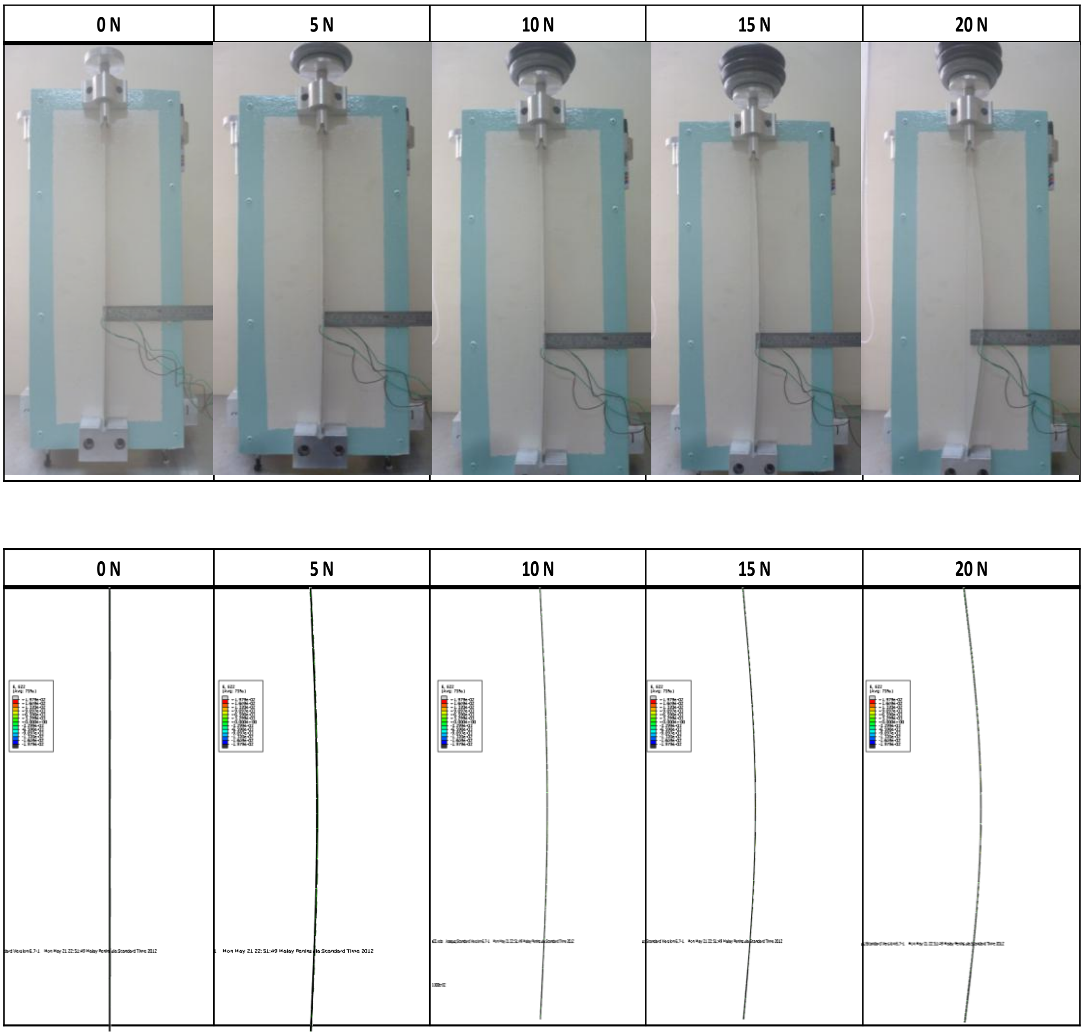


Figure 4.1: Comparison Physical Behavior of Buckle for Different Load Apply (Experimental and Simulation)

During run the experiment, the physical behavior of the buckle shape was recorded for each load from 0 N to 20 N. On the same time , the value of plain strain and the maximum displacement for buckle shape were collected.

From the Figure 4.1, it is show the comparison on physical behavior of the buckle shape from the experimental and simulation.. It is show that for every addition of load will increases the specimen's buckle shape and also will increase the displacement of buckle point. The specimen is easily buckle because of the end condition which is both end are pinned.

From the observation, the physical behavior of the model from simulation when buckle is same with the experimental model. Besides, it also show that the buckle shape of the model had follow the Euler's rule of buckling shape. Hence, from the experiment, we can conclude that comparison of physical behavior of buckle between experimental and simulation is approximately the same.

4.2.1 Result of Strain

Table 4.1: Data Strain (Experimental)

Load (N)	**Strain**			**Average**
	Test 1	**Test 2**	**Test 3**	
0	-0.000161	-0.000042	-0.000001	-0.000071
5	-0.000127	0.000027	0.000077	-0.0000375
10	-0.000056	0.000159	0.00023	0.000111
15	0.000085	0.000514	0.000553	0.000639
20	0.002584	0.002509	0.002439	0.0025

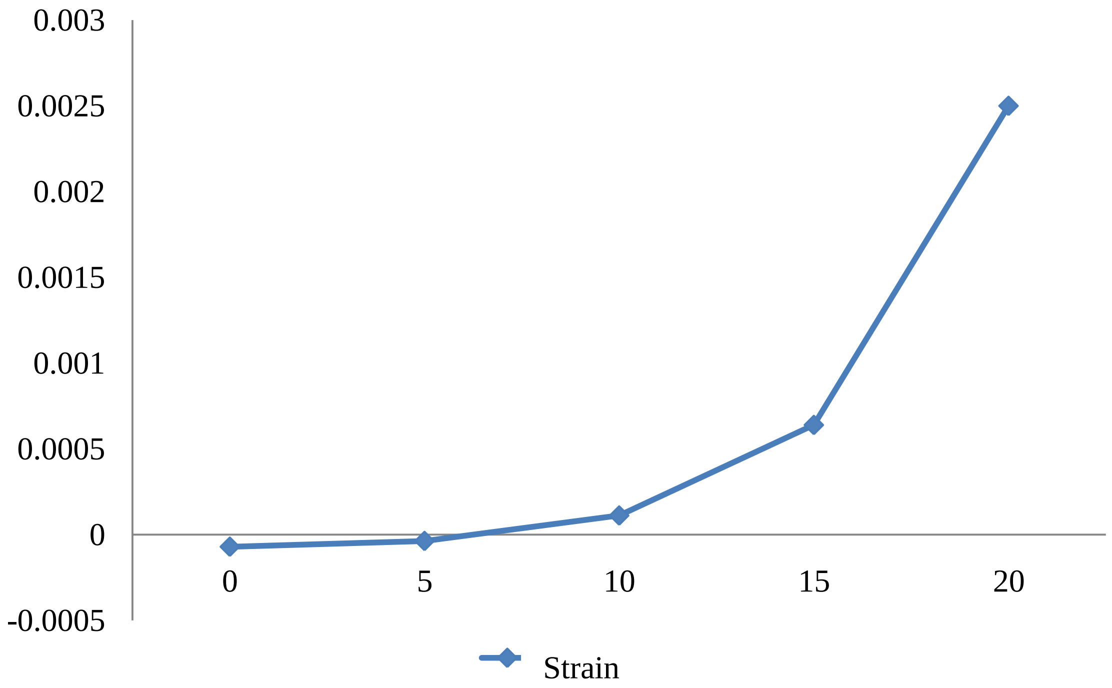


Figure 4.2: The Strain Versus Load (N)

From the graph in Figure 4.2, it show that the value of strain is increase when the load applied are increase. This is because due to deformation, when the load are applied, the specimen will be pressing the opposite edges of a sheet metal towards to one another. For small loads the process is elastic since buckling displacements disappear when the load is removed. However, when the load are not applied, which is 0 N, the value of strain is negative, it is because the specimen is under compressive state and it is not slightly straight due to weight of upper holder.

When the first until fourth load are applied, the specimen buckle is increase proportional to the load applied. On the fourth load, when the load is removed the specimen will returns to its original places and shape. At this point, the specimen still did not reach its critical load and it is in elastic region.

As the load is increase on the fifth load, the specimen begin to go undergo nonlinear elastic deformation at a stress called the proportional limit. At that point, the stress and strain no longer proportional as they were in the linear elastic region, but they are still return to the origin place and shape when the load is release. Permanent (plastic) deformation occurs when yield stress, Y of the material and critical load is reach. At this point, the specimen exceed its critical load and it is under plastic region. The critical load can calculate by using Euler rule, for the this experiment, the critical load is 19.7 N.

There are difference on the fourth and the fifth load because of the bending moments from the specimen. When the specimen's buckling deflection is high, the moments also became high. It is due to equation of moments, which is M= D x F, where M is moment, D is displacement and F is force.

4.2.2 Result of Displacement

Table 4.2: Data Displacement (Experimental)

Load (N)	**Displacement (mm)**			**Average**
	Test 1	**Test 2**	**Test 3**	
0	0	0	0	0
5	0.5	4.0	3.5	2.67
10	1	7.5	10	6.17
15	6.5	18.0	20	14.83
20	77	75	75	75.67

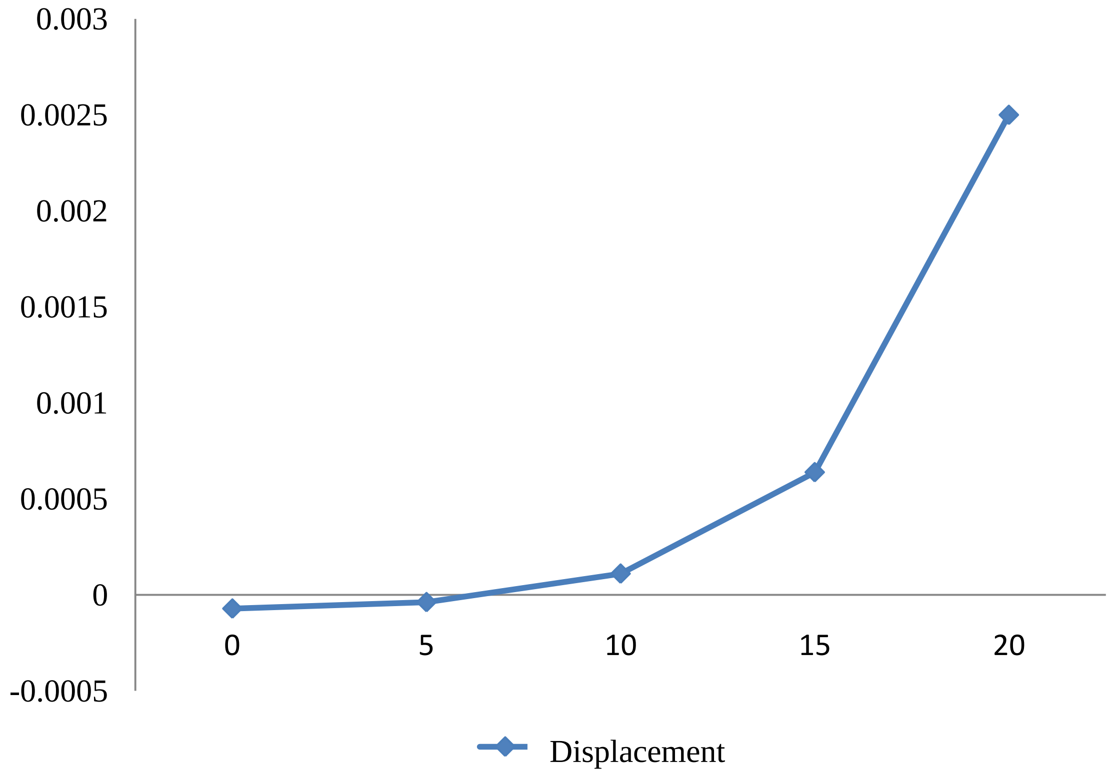


Figure 4.3: The Displacement (mm) Versus Load (N)

In Figure 4.3, it also show that the value of displacement of buckle also increase when we apply addition of load. It is linearly increase from 0 N to 20 N. However, on 20 N, the displacement of the buckle is highly increase compares to 0 N to 15 N. This is because of the specimen effective length factor, its depends on the conditions of end supports of the specimen. For this experiment, the conditions is both end are pinned and the factor is, k = 1.0.

For both end are pinned, the load that required to buckle is more lower than the condition for both end are fixed. On 0 N to 15 N, the specimen are still in beyond the elastic region. However, after the specimen exceed more than its critical load, it will give highly deformation displacement and this region is called the plastic region because within this range, if the force is removed, the specimen will not return to its original dimensions. If the specimen are applied more addition load, the specimen will be fracture.

4.3 ABAQUS Modeling

This Figure 4.5 show that value of strain during buckle under both end are pinned condition. The maximum value of strain for this model is 0.01979 at point node of 464. The positive value is indicate that the specimen is under tensile and negative value is under compressive. The red color is indicate the critical region of this model and it have a high strain value and for blue color it is vice versa. The red region indicate that the fracture will occur at that area if there are more additional load. By using simulation, the critical region of specimen can be predict.

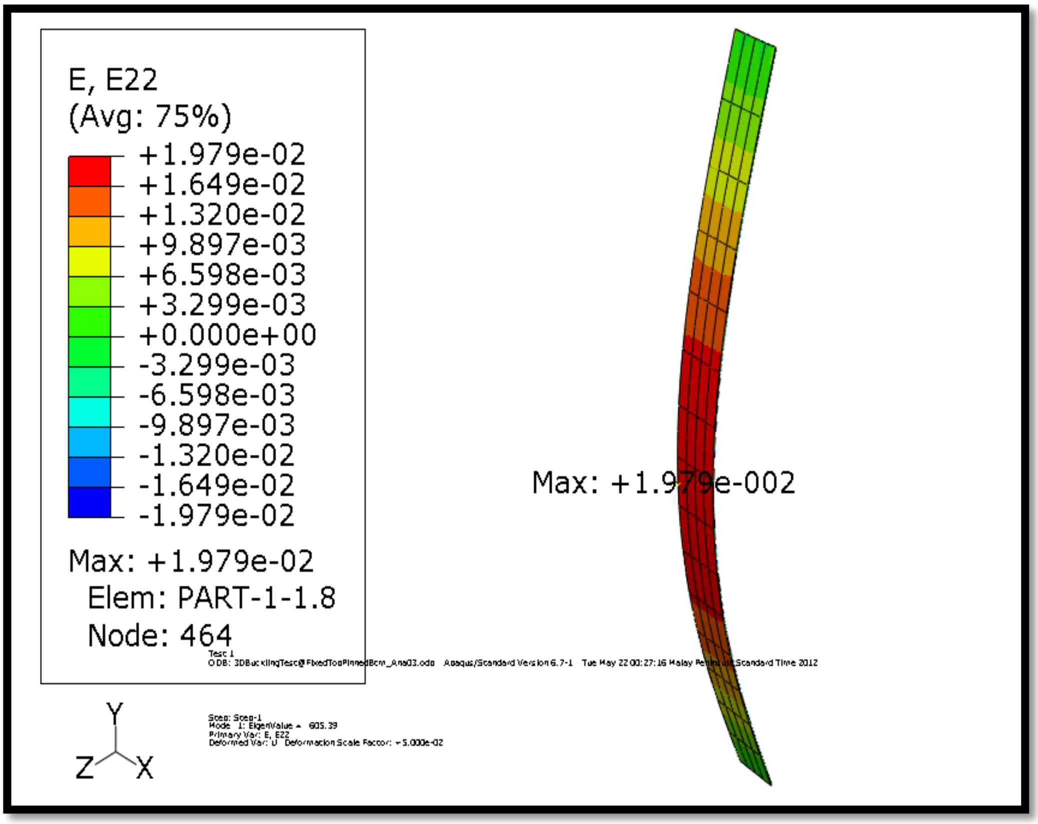


Figure 4.4: Simulation Model

4.4 Result of Comparison (Displacement)

Table 4.3: Comparison Displacement Between Experimental and Simulation

Load (N)	Experimental (mm)	Simulation (mm)	Percent Difference (%)
5	2.67	2.1	-27.14
10	6.17	5.7	-8.25
15	14.83	12.7	-16.77
20	75.67	43.8	-72.76

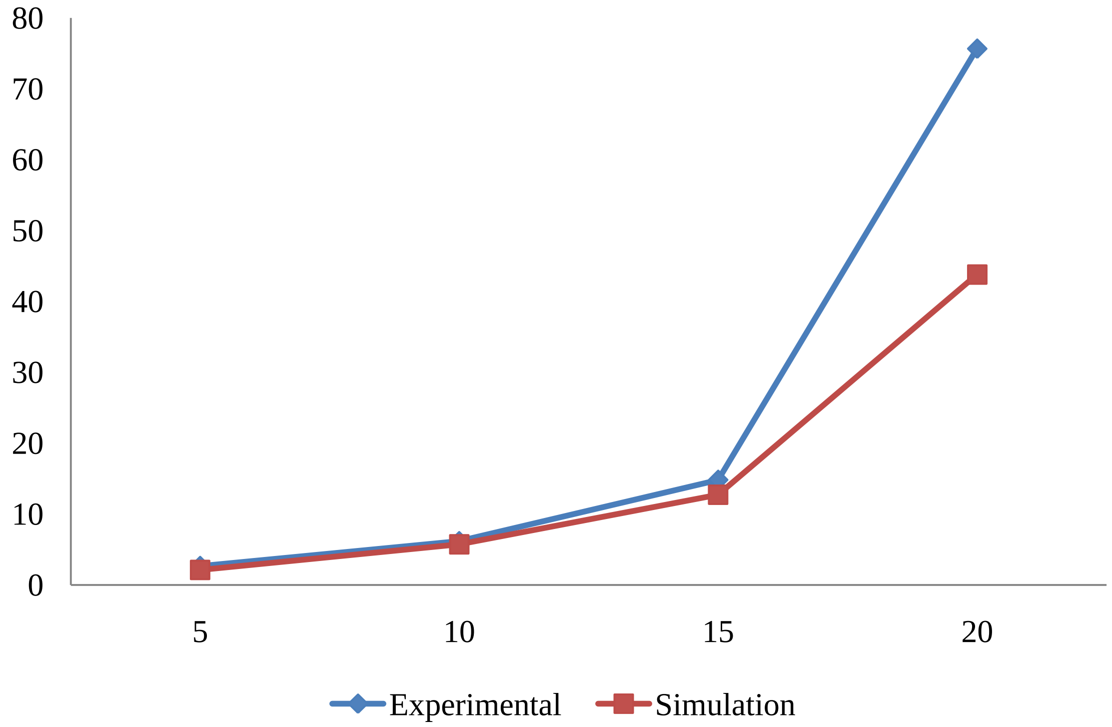


Figure 4.5: The displacement (mm) versus load (N)

Figure 4.5 shows that the comparison of displacement between experimental and simulation by using ABAQUS version 6.7 software. For every single load apply, the value of displacement for experimental and simulation give a different value. The highest percentage between experimental and simulation were 72.76% at the point of 20 N and the lowest percentage is 8.25% at the point of 10 N load are apply.

The value from experimental and simulation are not similar because there are some errors during the experiment. For each load apply, the value of displacement from the experimental is slightly vary with the simulation result. The error is occur because of many factors, the one of it is come out from simulation modeling. When using FEM, there are a lot of factors that need to consider and focused during build the modeling, for examples is the type of meshing, defines the materials properties, coating and base values, friction and iteration of model.

CHAPTER 5

CONCLUSION

5.1 Introduction

Chapter 5 summarizes all the main research points of this dissertation. It concludes that all the important information and observation from the research. Recommendation for future research is also included.

5.2 Conclusion

From the project that we have done, the buckling test rig that have been fabricate is validate and can be used in experiment lab. It is because its show the same physical behavior with the computer simulation during buckle and follow the Euler's rule. The pinned condition have make the specimen cannot move on x, y and z-axis but can rotate at any axis. From the result, it show that when we apply the load, the specimen will be buckle and if it exceed its critical load, it will enter plastic region and its shape cannot return to its original.

Besides that, the load and displacement point are directly proportionally. However, at certain load, there are highly increasing value of strain and displacement. It is due to moment which make the specimen more buckle. Furthermore, by using ABAQUS software, it is give the comparison of the displacement value from the experimental and simulation. The highest percentage of differences between displacements values are 72.76%, it is because of the simulation do not have a equation for determine the plain strain under surface. It is also because of the not very details on made a model on ABAQUS software. ABAQUS software for finite element analysis

(FEA) is known for its high performance, quality and ability to solve more kinds of challenging simulations than any other software.

On this project, we have learn and knowledge on how to handling machines and tools, by design and fabricate the buckling test rig. Then, we also have learn on project management and team works by distribute works among team members. Besides that, we also gain a knowledge on proper design and how to conduct different software with different experiment. For example finite element method must be done by using ABAQUS Software while strain gauge must be done by using DASYLab software and the design by using SOLIDWORKS 2011.

5.3 Future Research Recommendations

Based on the study carried out for this project., there are several recommendations in order to improve the result and product in the future. Below are the recommendations for the future work:

1) There need to more focused on modeling the model in ABAQUS because a lot of criteria that need to be details when make a model. For example, the value of friction on surface, poison ratio, heat transfer and many more should be very detail if we want to verify the result with experimental.

2) If this product can be portable, it can be commercialize for school and education institute .

REFERENCES

ANSYS User's Manual for Rev. 5.4, v. III - Elements Manual, 1995.

Budgen, N. F. 1947. Aluminum And Its Alloys. 2nd Edition. London: Sir Isaac Pitman & Sons Ltd.

Ben Young *, Jintang Yan, Finite element analysis and design of fixed-ended plain channel columns, 2002.

Ciarlet, P. and Lions, J. (1995). Handbook of Numerical Analysis : Finite Element Methods (P.1), Numerical Methods for Solids (P.2).

Concepts and Applications of Finite Element Analysis, 3rd ed.; Cook, Robert D./Plesha, Michael E./Malkus, David S.; John Wiley & Sons; 1989.

Dr. Burl E. Dishongh "Paradigm for Teaching Structural Technology". Journal of Construction Education, Spring 2003, Vol. 8, No.1, pp 28-37 .

D. B. Stephen, G.P. Steven. Error estimation for plate vibration elements, Research report, FEARC-9408, Finite Element Analysis Research Centre, University of Sydney (1994).

David Roylance . "Finite Element Analysis", February 28, 2001.

E. O. Doebelin, Measurement Systems:- Application and design, 5th ed., "Theory and Practice of strain gages McGraw Hill", 2008

En Ye,Chang Lin and Jennifer A.Palack Wahl. 2007. Enhancing software engineering education using teaching aids 3D online virtual words.

Ever J Barbeno, Edgar K Dede and Shannon Jones. 1999. Verification of buckling mode interaction in intermediate length composite column.

Foudil Mohri, Cherif Bouzerira, Michel Potier-Ferry. "Lateral buckling of thin-walled beam-strip elements under combined axial and bending loads", 20 October 2006;

Fatimah Denan, Mohd Hanim Osman and Sarrifuddin Saad. 2010. The study of lateral torsional buckling behavior of beam with trapezoid web steel section by experimental and finite element analysis.

Georges CAILLETAUD & Saber EL AREM Introduction to Finite Element Method, 2010.

Jerome M. Solberg*, Panayiotis Papadopoulos, A Finite element method for contact/impact, 1998.

J.E. Akin, "FEA Buckling Analysis" 2009.

Karl Hoffmann, An Introduction on Measurements using Strain Gages 1989.

Kalpakjian, S. and Schmid, S. 2006. Manufacturing Engineering and Technology 5th Edition. Singapore: Pearson Education South Asia Limited

L.A. Dobrzański, A. Śliwa*, T. Tański, Finite Element Method application for modelling of mechanical properties,2009.

Mohd Razi. 2007. Buckling behavior of thin walled angle section beam subjected to pure bending moment.

R. Kopp, M.L. Cao and M.M. Souza"Muti-level simulation of metal forming process", Proc. 2nd ITCP, Stuttgart, August 1987, pp. 1128-1234.

Osman 2004; Marcus et al., n.d.; Lira et al.

Shigley, "Shigley's Mechanical Engineering Design", 8th edition,2006, pp. 177-178.

Sérgio Tonini Button. "Numerical Simulation and Physical Modeling as Educational Tools to Teach Metal Forming Processes"

S. Ju, C. Fan, G. Wu, Three-dimensional finite element soft steel Boltem connections, Engineering Structures 26 (2004) 403-413.

Suresh R. Dash , Subhamoy Bhattacharya , Anthony Blakeborough" Bending–buckling interaction as a failure mechanism of piles in liquefiable soils"2009.

Tokyo Sokki Kenkyujo. 2008. Developing Strain Gauge and Instrument. Smith, W. F. 1996. Principle of Materials Science and Engineering. 3rd Edition. New York: McGraw-Hill.

Philip, A. and Schweitzer, P.E. 2003. Metallic Material: Physicals, Mechanical, and Corrosion Properties. New York: Marcel Dekker Inc.

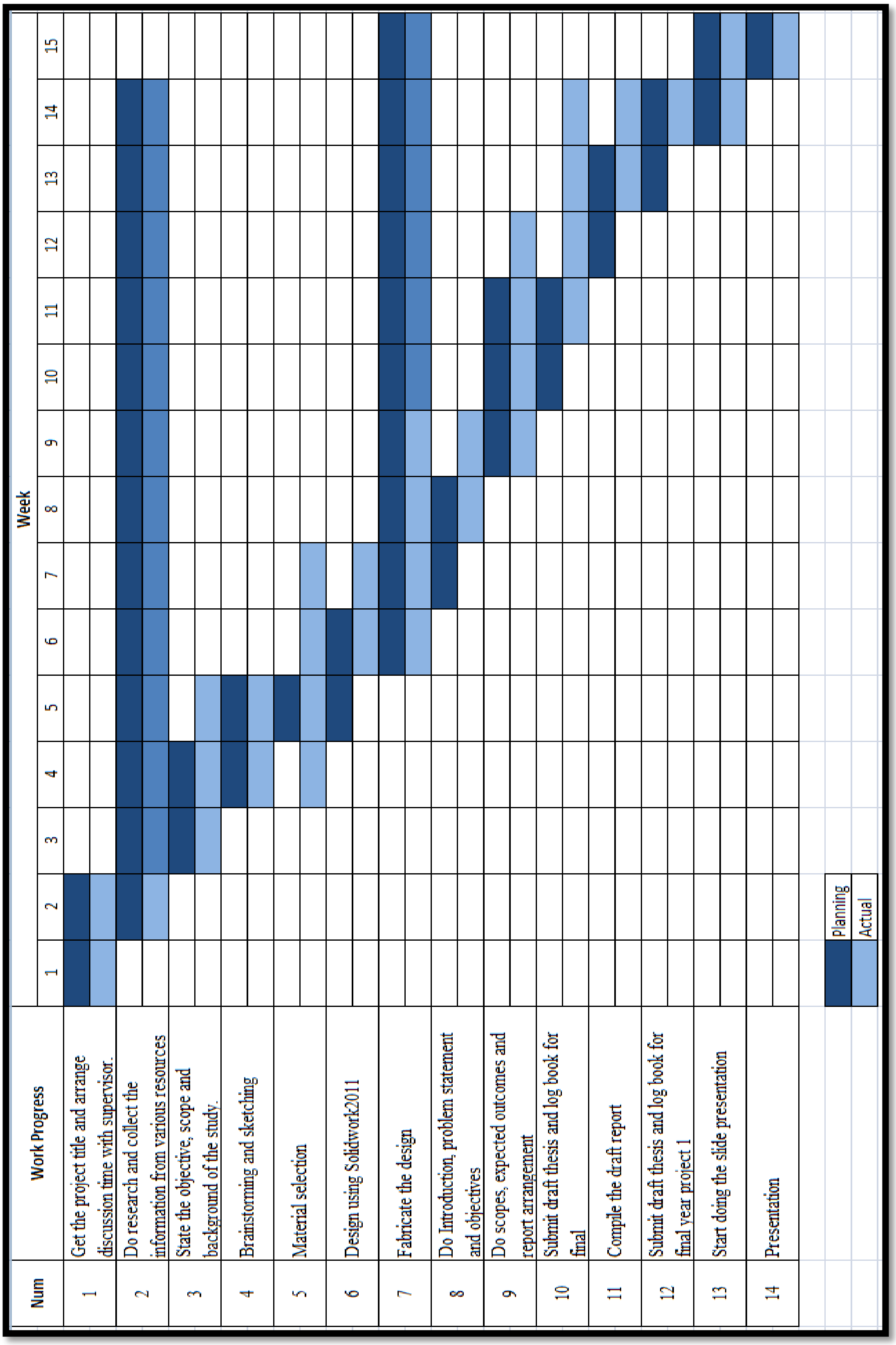
William, F.S. and Javad, H. 2003. Foundations of Materials Science and Engineering. New York: McGraw-Hill Professional.

Y.H. Moon and C.J. Van Tyne. 2000. Validation via FEM and plasticine modeling of upper bound criteria of a process-induced side-surface defect in forgings, Journal of Materials Processing Technology, Vol. 99, pp. 185-196.

Z.Ahmad@Manap, M.R.M Rejab, M.M Norr, R.Daud, M.S.M Sani and K.Kadirgama. " Effect of Adverse Environment on Buckling of Composite Laminates". MUCEET2009.

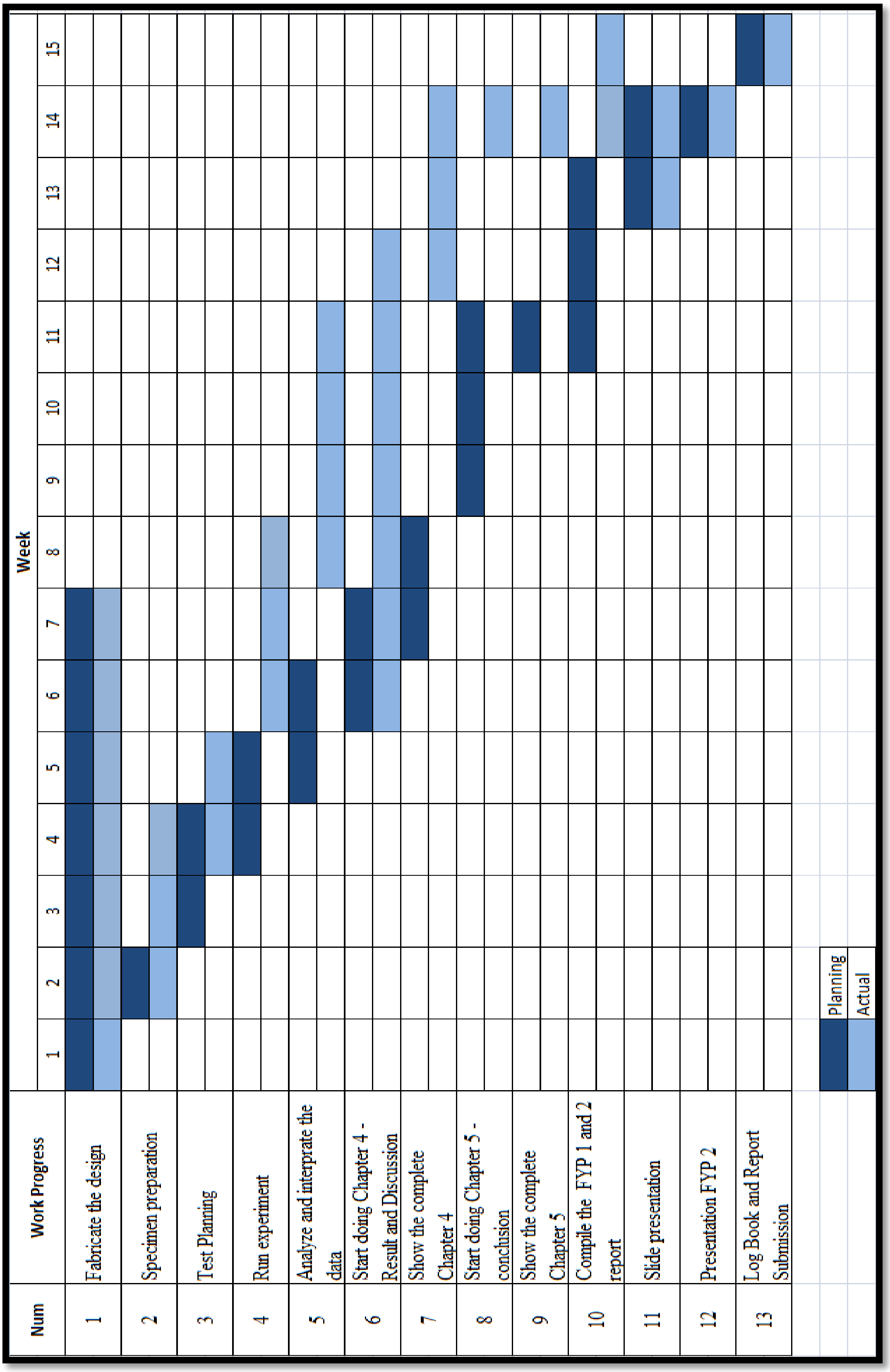
APPENDIX A1

GANTT CHART FOR FINAL YEAR PROJECT 1



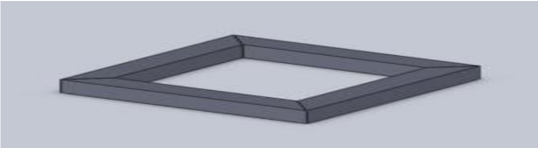
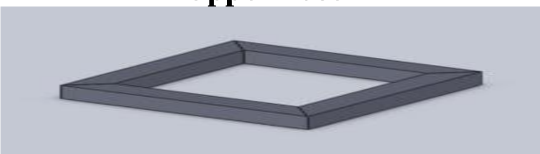
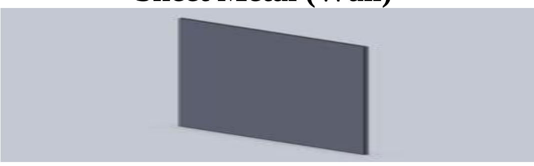
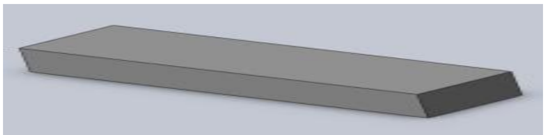
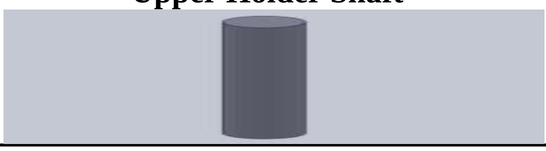
APPENDIX A2

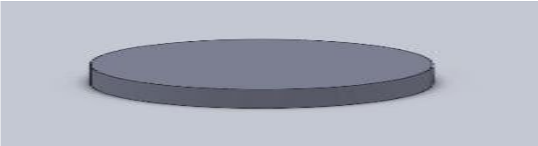
GANTT CHART FOR FINAL YEAR PROJECT 2



APPENDIX B1

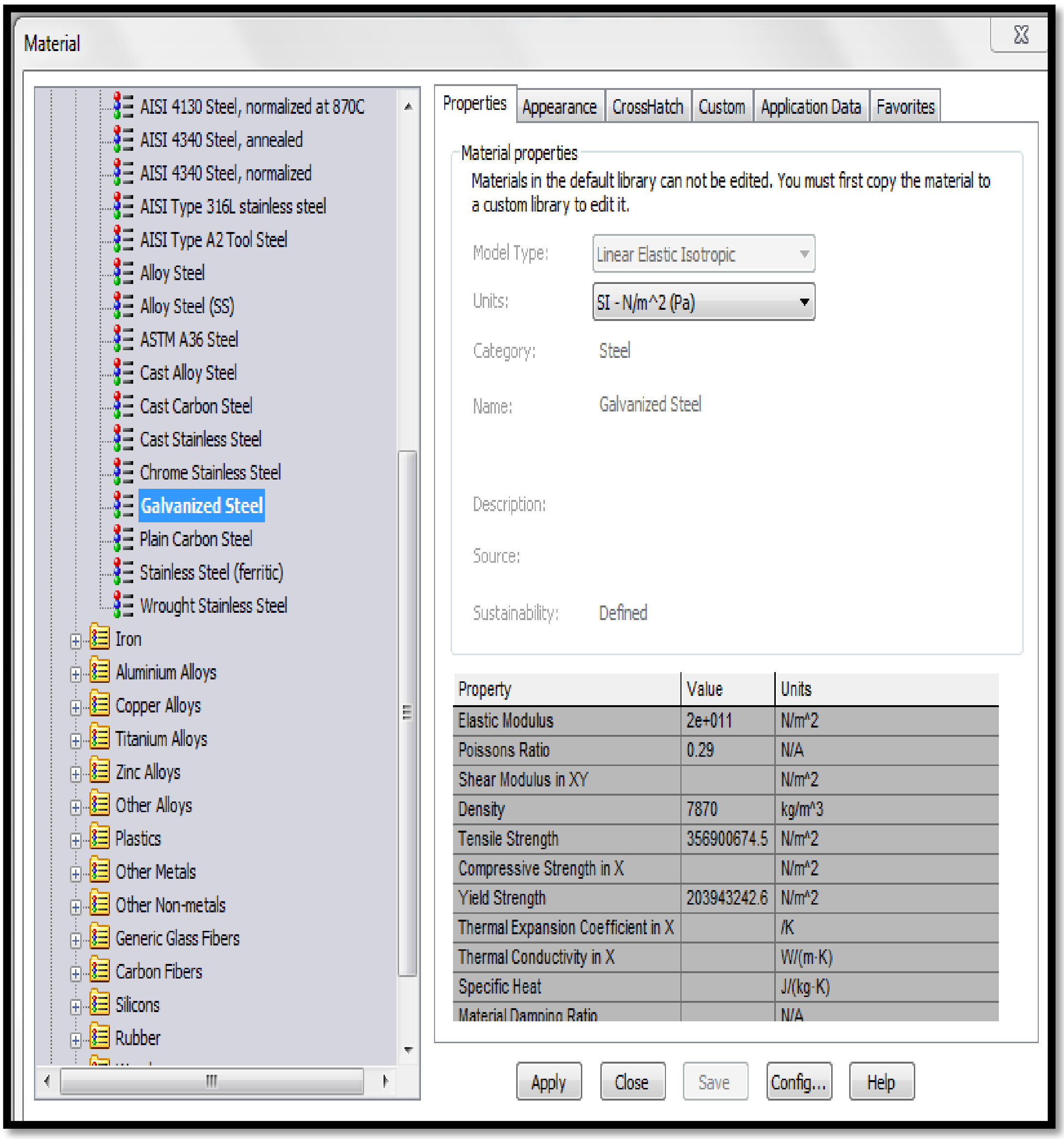
DESIGN COMPONENT OF TEST EQUIPMENT (3D)

Component	Material	Dimension (mm x mm x mm)	Machine Process
Lower Base 	Rectangular Hollow	50 x 50 x 400	Disc Cutter Welding
Upper Base 	Rectangular Hollow	50 x 50 x 400	Disc Cutter Welding
Side Support 	L-shape steel	25 x 25 x 800	Disc Cutter Welding
Sheet Metal (Wall) 	GI steel	400 x 725 x 3	Shearing Machine
Upper Holder 	Solid Aluminum	35 x 35 x 150	Band saw Milling CNC machine
Lower Holder 	Solid Aluminum	50 x 50 x 150	Band saw Milling
Upper Holder Shaft 	Solid Aluminum	D=15 t=100	Turning

Load Holder 	Aluminum	D=80 t=10	Turning Milling

APPENDIX B2

MODELING PROPERTIES (SOLIDWORKS 2011)



APPENDIX B3

EULER RULE

Based on Euler strip formula, we can determine the value of critical force for our specimen before the experimental test start.

Calculation for the Euler strip formula,

$$P_{cr} = \frac{C\pi^2 EI}{l^2}$$

where is,

P_{cr} = Critical force
C = Constant depend on ends condition (C = 1, for ends pinned)
E = Elongation
I =Moment of Inertia
l =Length

$$P_{cr} \quad = \frac{1 \text{ x } \pi^2 \text{ x } 207e^9\text{Pa x } \frac{1}{12} \text{ x } 0.03\text{m x } 0.001^3 m}{0.5^2\ m}$$

= 19.73 N , 1.97 kg