

Engineering Investigation on Large Deformation
Properties of a Thermoplastic Sheet

TAN YIK LONG

FB12050

Report submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Engineering in Manufacturing Engineering

Faculty of Manufacturing Engineering

UNIVERSITI MALAYSIA PAHANG

May 2016

SUPERVISOR'S DECLARATION

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

Signature :

Name of supervisor : Associate Professor Dr Abdul Aziz Bin Jaafar

Position : Lecturer of UMP

Date : 26/5/2016

STUDENT'S DECLARATION

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

Signature :

Name : TAN YIK LONG

ID Number : FB12050

Date : 26/5/2016

ACKNOLEDEGEMENT

Firstly, I am grateful to the God for giving me strength, inspiration and also the good health that were necessary throughout the accomplishment of this thesis. Hereby, I would like to express my sincere gratitude to my supervisor Associate Professor Dr. Abdul Aziz Bin Jaafar of Faculty of Manufacturing Engineering (FKP) for the continuous support during the research and also his motivation, patience and immerse knowledge. His guidance helped me a lot during the research period and also when I am writing the thesis. The research will not be successful without the assistant of him. Last but not least, I would like to thank my beloved family and friends for their encouragement and support throughout the period of this research.

ABSTRACT

The use of structural calculation software based on finite-element analysis is nowadays a common practice when designing new industrial products. In order to make an adequate prediction of the service behavior of plastic components, it is necessary to carry out appropriate analysis when working with the software. ANSYS Parametric Design Language (APDL) is an important tool to perform finite element analysis to predict how the product will react in the real world situation. This research paper is focused on to investigate deformation properties of thermoplastic sheet. The investigation will be done through simulation approach, which is by using ANSYS APDL to perform the finite element analysis. The material that is going to be tested is polypropylene, and the testing standard will be followed the standard ASTM D638. The simulation results will be collected and interpreted to determine the de-formation properties and the service behavior of the material.

ABSTRAK

Penggunaan perisian pengiraan struktur berdasarkan analisis unsur terhingga adalah amalan biasa pada masa kini apabila mereka membentuk produk industri baru. Dalam usaha untuk membuat ramalan sifat-sifat komponen plastik, adalah perlu untuk menjalankan analisis yang sesuai dengan penggunaan perisian. ANSYS Parameter Design Language (APDL) adalah alat yang penting untuk melaksanakan analisis unsur terhingga untuk meramalkan bagaimana produk itu akan bertindak balas dalam keadaan dunia sebenar. kertas kajian ini adalah memberi tumpuan kepada untuk menyiasat ciri-ciri ubah bentuk lembaran termoplastik. Siasatan akan dilakukan melalui cara simulasi, iaitu dengan menggunakan ANSYS APDL untuk melaksanakan analisis unsur terhingga. Bahan yang akan diuji adalah polypropylene, dan standard ujian akan diikuti ASTM D638 standard. Keputusan simulasi akan dikumpulkan dan ditafsirkan untuk menentukan sifat-sifat ubah bentuk dan tingkah laku perkhidmatan yang bahan.

TABLE OF CONTENTS

	Page
SUPERVISOR'S DECLARATION	iii
STUDENT'S DECLARATION	iv
ACKNOWLEDGEMENT	v
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xi
LIST OF FIGURES	xii
LIST OF ABBREVIATIONS	xviii
CHAPTER 1 INTRODUCTION	
1.1 Introduction	1
1.2 Problem Statement	3
1.3 Method to Solve	3
1.4 Hypothesis	3
1.5 Objective	4
1.6 Project Scope	4
CHAPTER 2 LITERATURE REVIEW	
2.1 Introduction	5

2.2	Finite Element Analysis	
2.3	Dog Bone Shape Testing Specimen	8
2.4	Tensile Properties	11
	2.4.1 Linear Elastic Region	13
	2.4.2 Yield Point	14
	2.4.3 Ultimate Tensile Strength	15
2.5	Polymer	19
	2.5.1 Elastomer	21
	2.5.2 Thermosetting Polymer	22
	2.5.3 Thermoplastic Polymer	23
2.6	Summary	24

CHAPTER 3 METHODOLOGY

3.1	Introduction	26
3.2	Process Flow Chart	27
	3.2.1 Project Descriptions	28
3.3	Material and Software	29
	3.3.1 Dog Bone Shape Testing Specimen	29
	3.3.2 ANSYS Parametric Design Language	30
3.4	Procedures To Set Up ANSYS	29

CHAPTER 4 RESULTS AND DISCUSSIONS

4.1	Introduction	5
4.2	Uniaxial Stress Strain Analysis	
4.3	The Stress Strain Curve	8

CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS

5.1	Introduction	5
5.2	Conclusions	
5.3	Recommendations	8
	REFERENCES	35
	APPENDICES	38
A	Gantt Chart	39

LIST OF TABLES

Table No.	Title	Page
4.1	Input summary of element shell 63	56
4.2	Input summary of element shell 63 (cont)	57
4.3	List of stress and strain	61

LIST OF FIGURES

Figure No.	Title	Page
2.1	Comparisons of Simulation Result with Experimental Result for Bending Test in Polypropylene	7
2.2	Comparisons of Simulation Result with Experimental Result for Penetration Test in Polypropylene	8
2.3	Comparisons of Simulation Result with Experimental Result for Bending Test in Polycarbonate	8
2.4	Comparisons of Simulation Result with Experimental Result for Penetration Test in Polycarbonate	9
2.5	Original State of Dog Bone Shape Testing Specimen	10
2.6	Broken State of Dog Bone Shape Testing Specimen	10
2.7	Expected Broken Place of Dog Bone Shape Testing Specimen	11
2.8	Proper Alignment between the Specimen Axis and Pull Direction	12
2.9	Improper Alignment between the Specimen Axis and	13

Pull Direction

2.10	Stress Strain Curve	13
2.11	Comparison of Stress Strain Curve between a Brittle Material and also a Ductile Material	15
2.12	Schematic Illustration of Linear Polymer	16
2.13	Schematic Illustration of Branched Polymer	17
2.14	Schematic Illustration of Cross-linked Polymer	17
2.15	Schematic Illustration of Network Polymer	18
2.16	Elastomer in Unstressed State	20
2.17	Elastomer in Stressed State	20
2.18	Schematic Molecular Structure of the Thermosetting Polymer	21
2.19	Schematic Molecular Structure of the Thermoplastic Polymer	22

3.1	Flow Chart	24
3.2	Dimension of Dog Bone Shape Testing Specimen	26
3.3	Logo of ANSYS	27
3.4	Drawing of Dog Bone Shaped Specimen	28
3.5	Dimension of Dog Bone Shape Testing Specimen	29
3.6	Creation of Area	29
3.7	Small Window of Create Area by Lines	30
3.8	Selection of the Lines	30
3.9	Area of the Model	31
3.10	Selection of Element Type	32
3.11	Element Type of Shell 63	33
3.12	Defined of Real Constant	34
3.13	Window of Element Type for Real Constant	35

3.14	Set Up of Thickness	36
3.15	Set Up of Material Properties	36
3.16	Set Up of Young's Modulus and Poison Ratio	37
3.17	Mesh Tool	37
3.18	Checking of Meshing Attributes	38
3.19	Selection of Both Side Line	39
3.20	Setting of Element Size on Both Sides	39
3.21	Selection of Line at the Grip Section	40
3.22	Setting of Element Size on Grip Section	40
3.23	Selection of Curve Line	41
3.24	Setting of Element Size on Curve Line	41
3.25	Selection of Middle Line	42
3.26	Setting of Element Size on Middle Line	42

3.27	Window of Mesh Area	43
3.28	Selection of Area	43
3.29	Result of Meshing Process	44
3.30	Applying Structural Load	44
3.31	Defined of Displacement Load	45
3.32	Defined of Degree of Freedom	46
3.33	Result of Displacement Loading	46
3.34	Defined of Force	47
3.35	Setting of Force	48
3.36	Result of Force Applied	48
3.37	Window of Begin Solution	49
3.38	Window of Solution is Done	49
3.39	Window of Nodal Solution	50

3.40	Analysis Results	51
3.41	Window of Time History Variable	51
3.42	Window of Adding Nodes Data	52
3.43	Window of Choosing Nodes	52
3.44	Results of Stress Strain Curve	53
4.1	Dimension of testing specimen ASTM D638	55
4.2	Element shell 63	55
4.3	The uniaxial stress strain tensile simulation result	57
4.4	The stress strain curve	59
4.5	The interpretation of stress strain curve	59
4.6	Resilience and toughness of material	62

LIST OF ABBREVIATIONS

FEA Finite Element Analysis

DMA Dynamic Mechanical Analyzer

APDL ANSYS Parametric Design Language

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

The prompt development of modern process industries has caused the information about properties of materials to become a more important issue to know, especially for polymeric materials. Polymers are usually divided into two groups, which are thermosets and also thermoplastics. The difference between thermosets and thermoplastics is that they have different mechanical properties and the manner of forming. Thermoplastics contributed 90% of the produced plastics in the world and most of them were used in the automotive industry. The reason for this is because thermoplastics are a more affordable and easily machinable material. [1]

Recently, polymeric materials, especially thermoplastic materials, are more frequently used in the automotive sector. Thermoplastic materials are usually applied in metallic components, passenger components in a car. In the automotive industry, the use of plastic has become more and more important. In the past, plastic was regarded as a low-quality material in people's perception, due to that, it was only used to build some simple mechanisms and basic applications such as non-loading applications inside the car. In fact, plastics are a lot better if compared to metal in terms of cost, freedom in design,

resistance, formability and also environmental friendly. With the advancement of technology, we tend to see more plastic has dramatically increased their usage in modern car. Plastics are able to offer good mechanical properties together with the outstanding appearance and also possible to self-coloring. Nowadays, plastic is also used improve the energy efficient of a car by reducing the car overall weight and at the same time, it providing good durability, toughness, corrosion resistance, design flexibility and excellent performance at a lower material cost. Therefore, plastic has increased its application in modern car because it is able to offer good quality product even at a lower cost [2,3].

In order to produce plastics that are able to optimize the design of the product to fulfill the client's needs before the design is implemented in the production line, the structural analysis of plastic material is essential. There are a lot of methods and tools able to perform the structural analysis of plastic material. Finite element analysis is one of the methods to perform structural analysis on material. Finite element analysis (FEA) is a computerized approach to find out how a product will behave when it faces the real world forces, heat, vibration and so on. The product may break, wear out, or perform well according to what it is designed under finite element analysis. Although it is called as analysis, but it is used to carry out the prediction on the product of what is going to happen when the product is being used in real world. Besides that, its purpose also included reduces the manufacturing cost, labor cost and also improve the product quality. ANSYS Parametric Design Language (APDL) is a tool designed to perform finite element analysis. It allows users analyze and obtain the desired results by building a finite element model in it. Therefore, ANSYS Parametric Design Language (APDL) is important to perform finite element analysis to predict how the product will react in the real world situation. [4]

This research paper is focus on to investigate deformation properties of thermoplastic sheet. The investigation will be done through simulation approach, which is by using ANSYS to perform the finite element analysis.

1.2 PROBLEM STATEMENT

Engineering approach to determine the deformation properties and the service behavior of the thermoplastic sheet.

1.5 OBJECTIVE

The objectives of this project are:

1. To investigate the tensile deformation properties of thermoplastic sheet.
2. To determine the mechanical behavior of the thermoplastic sheet by using ANSYS APDL.
3. To perform uniaxial stress-strain analysis on thermoplastic sheet by using ANSYS APDL.

1.6 PROJECT SCOPE

The scopes of this project are:

1. Deformation properties of thermoplastic sheet.

2. Designation of the testing specimen.
3. Finite element analysis by using ANSYS APDL.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter provided a review of existing literature to support the study undertaken in this project. The sources of the literature come from journal and article, information from the website, reference book, lecture's slide and so on. This chapter will provide a better understanding on problem statement and also project scope of this project. Lastly, a discussion of literatures related to the deformation properties and service behavior of thermoplastics field is performed and some recommendations were also being proposed in this chapter.

2.2 FINITE ELEMENT ANALYSIS

Finite element analysis was developed in the year 1943 by a German mathematician Richard Courant. He used numerical analysis and minimization of variation calculus to get the approximate solutions to vibration system. Finite element analysis has been widely accepted as a common tool to perform structural modeling and

simulation. [5] Finite element analysis helps in determine whether a new design product can perform well in the real world situation.

Because of the tough competition, all industrial manufacturers must work hard to optimize the conflicting technical and financial goals although the increasing in product complexity. This challenge is going to be tough because of the randomness, therefore the use of probabilistic tools is important to solve the problem. There are two tools, namely the ANSYS Probabilistic Design System and the ANSYS DesignXplorer has been released by the ANSYS Inc to solve the situation. In the paper of “Probabilistic finite element analysis using ANSYS”, it describes the problems that can be addressed, the underlying algorithms implemented and methodologies of these methods in both tools. Besides that, the special topic of this paper is it included the discussion and also explanations of the Variational Technology which can be found in both of the tools. This technology is able to give an accurate and high order response surface based on the single finite element analysis. In this paper also discussed all the strength and weakness of these method. Besides that, the possibility to reduce the processing time using parallel computing is also been discussed in this paper. Different ways to get the accuracy of the results taken with the different probabilistic methods are also discussed in this paper. Finally, the method to optimize the possibly conflicting goals are also highlighted.

In the research in “Finite-element analysis of quasi-static characterization tests in thermoplastic material”, they have used software ANSYS to do the analysis of the service behavior of thermoplastic when working. They have used two types of thermoplastic which are polypropylene and polycarbonate for testing. In their research, they need to search a better correlation level between the simulation result which is done by using ANSYS and the experimental mechanical test results. There are two

quasi-static tests they carried out to make a comparison between the experimental result with the simulation result, first test is 3-point bending test and another is plate penetration test. [6] Figures 2.1, 2.2, 2.3, 2.4 below show the results they got for their research. Figure 2.1 showing the comparison for the bending test while figure 2.2 showing the comparison for the penetration test, both for the material polypropylene. The next two figures 2.3 and 2.4 show the same thing but for material polycarbonate. The conclusion that can be made for this research is the best correlation between simulation result and experimental result can be getting by using correct friction coefficient.

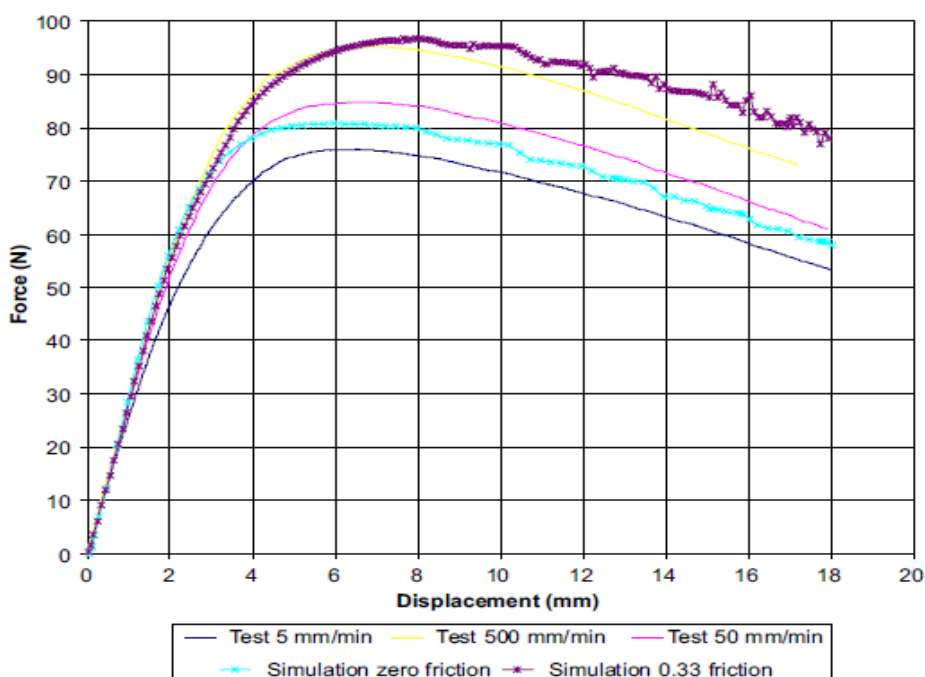


Figure 2.1: Comparisons of Simulation Result with Experimental Result for Bending Test in Polypropylene [6]

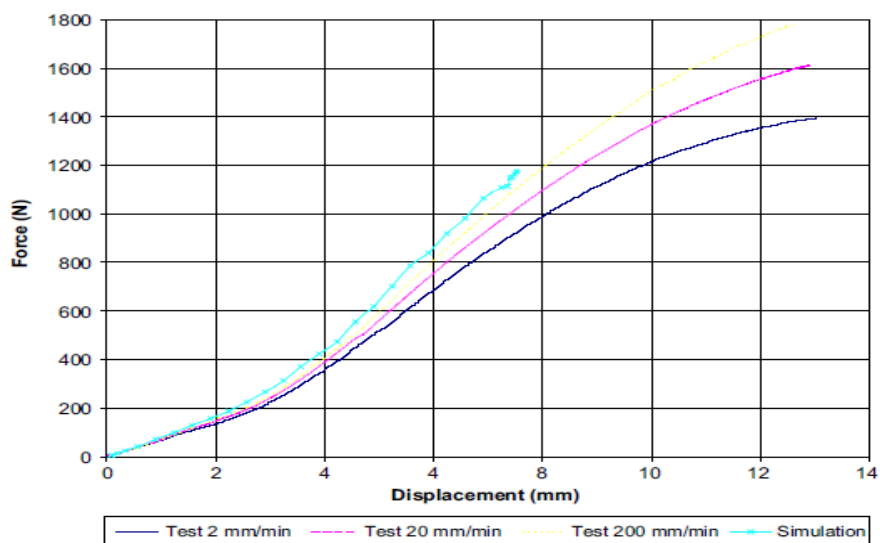


Figure 2.2: Comparisons of Simulation Result with Experimental Result for Penetration Test in Polypropylene [6]

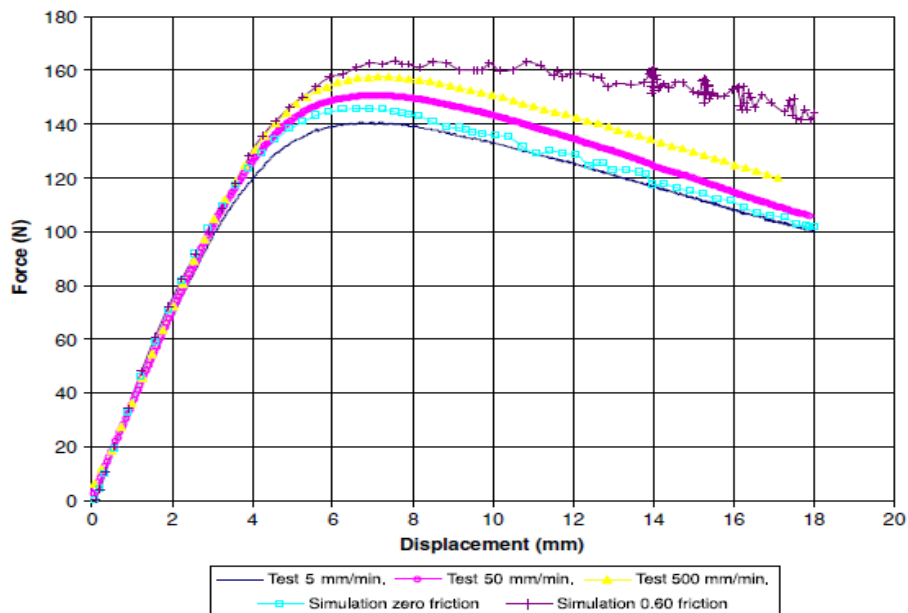


Figure 2.3: Comparisons of Simulation Result with Experimental Result for Bending Test in Polycarbonate [6]

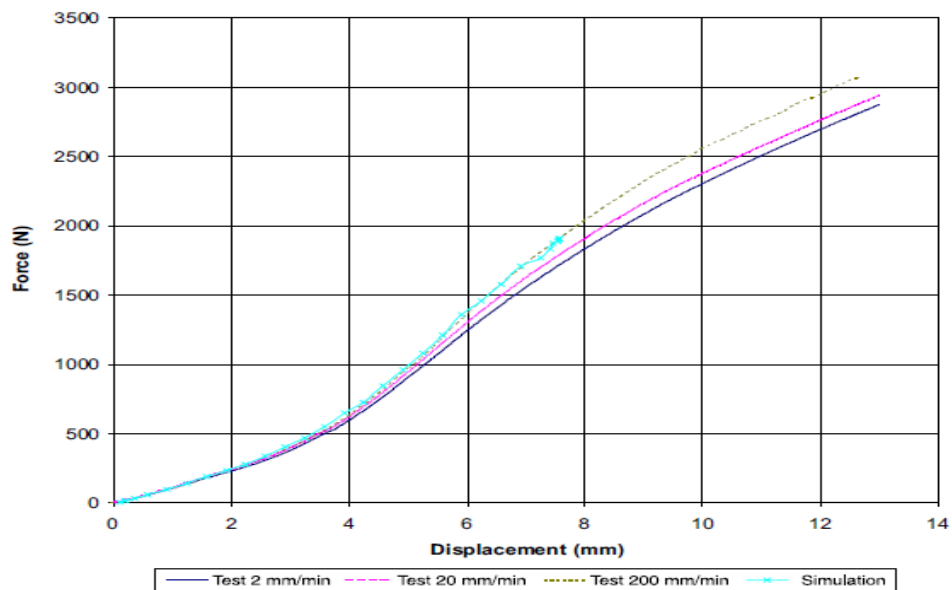


Figure 2.4: Comparisons of Simulation Result with Experimental Result for Penetration Test in Polycarbonate [6]

2.3 DOG BONE SHAPE TESTING SPECIMEN

Dog bone shape testing specimen is normally used in tensile testing. It has a gauge section in the middle and also two shoulders at both ends each. The sample has a wider shoulder than the middle gauge section. It is designed in this shape to make sure that when it is loaded with the tensile force, the stress concentration is happened at the middle gauge section. [9] Figure 2.5 and figure 2.6 below showing the dog bone shape testing specimen in original state and also in broken state respectively.



Figure 2.5: Original State of Dog Bone Shape Testing Specimen [10]



Figure 2.6: Broken State of Dog Bone Shape Testing Specimen [10]

The purpose of make sure the stress concentration is happened at the middle is important because this will ensures a higher chance that the sample will only break down at the middle gauge section. The testing material is reaching the maximum tensile strength limit when the rupture section is at the middle part of the testing specimen. On the other hand, if the rupture section is at the either end of the testing specimen, it is considered as failure of the testing and it may due to improper loading manner. So, the dog bone shape testing specimen is designed to make sure that the specimen will only rupture due to maximum tensile loading. Figure 2.7 below shows the failure location of testing specimen in the tensile strength testing.

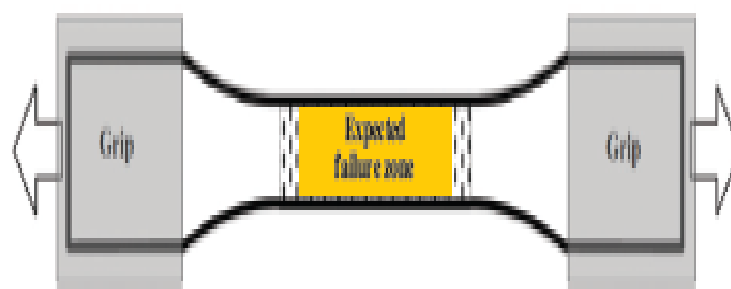


Figure 2.7: Expected Broken Place of Dog Bone Shape Testing Specimen [11]

There are two major methods used to prepare dog bone shape testing specimen, cutting and also molding. A die cutter is needed to cut the sample from the material, the shape and the dimension will follow the shape and the dimension of the blade. Preparing the testing specimen through molding method is more complicated and more expensive if compare with by using a die cutter. This process requires the testing material to be in soft flexible state and then poured it into the mold with the desired shape and dimension. After it is harden, then it is ready to be tested. The testing specimen prepared through molding method is usually stiffer than testing specimen prepared through die cutting process. The dog bone testing specimen can be made up of any material that can be tested in tension, but the most common material is plastics, polymer and rubber. Due to their ductile nature, it is easier to cut them through the sample material, so the cutting method is more preferred to be used to prepare the testing specimen from flexible material. [9]

2.4 TENSILE PROPERTIES

Tensile properties show how a material will react to the force when applying tension on it. Tensile test is a method used to find the tensile strength, elastic limit, young's modulus, yield point, yield strength, and so on. The test is usually carried out with the use of a universal testing machine. Tensile test is a mechanical test where the testing specimen is loaded with the proper way on the universal testing machine. Then the applied load is measured and also the elongation of specimen over certain distance. The testing specimen is usually in dog bone shape. This is because the dog bone shape testing specimen can make sure that the testing sample is rupture only due to maximum tensile loading. But sometimes, the testing specimen may rupture at the place outside of the middle gauge section, this is because the alignment between specimen axis and also the pull direction is not match. Other than that, over tightening at the grips part of the specimen may cause weakening of grip positions and then cause failure during testing process. [14] In overall, tensile properties may vary with different thickness of specimen, the method of preparation, and the testing speed. [12] Figure 2.8 and figure 2.9 below show the proper and improper alignment between the specimen axis and also the pull direction respectively.



Figure 2.8: Proper Alignment between the Specimen Axis and also Pull Direction [11]

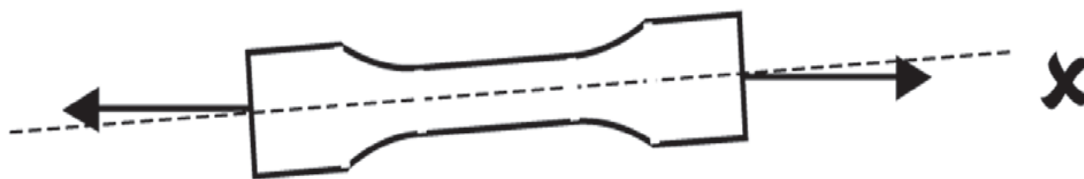


Figure 2.9: Improper Alignment between the Specimen Axis and also Pull Direction

[11]

The tensile test will produce a stress versus strain curve. Every material has its own stress strain curve and the curve is relating the applied stress to the resulting strain of every material. Figure 2.10 below shows the example of a stress strain curve. From the figure, we can see that the curve is increases continuously until it reaches its fracture point. [14]

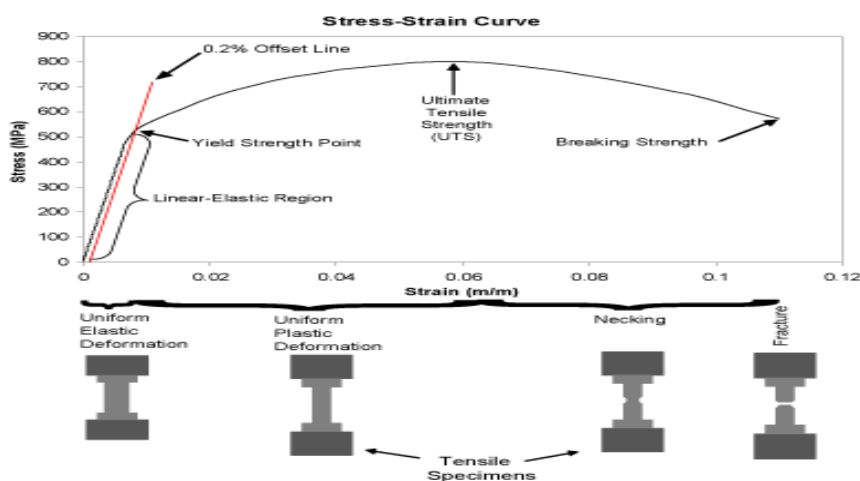


Figure 2.10: Stress Strain Curve [14]

2.4.1 Linear Elastic Region

From the figure 2.10, we can see in the linear elastic region, the stress is increase linearly with the strain. Since it is linear elastic portion, so there will be no plastic deformation occurred at this region. In this state, the material will tend to return to the original state when the stress is decreased. This region obeys the Hooke's law which indicates that the result of stress over strain is a constant. The gradient of the line in this linear elastic region is called modulus of elasticity or young's modulus. The young's modulus indicates the properties of a material when the material was applied with stress, deformed and when it goes back to original state after the applied stress is removed. In other word, it is a measurement of stiffness of a material. [13] In order to calculate young's modulus, we need to divide the stress by the strain in material. The unit of young's modulus will follow the unit of stress since the strain is without unit.

2.4.2 Yield Point

For ductile materials in stress strain curve, at certain point, the Hooke's law can no longer be applied due to the strain increases faster than the stress. At this point, there will be some permanent deformation happened at the testing specimen. The testing specimen will not return to its original state although the load has been removed. For brittle materials, there will be only a little or no plastic deformation at all, the brittle testing specimen will fracture after the linear elastic region in the stress strain curve. Below is a Figure 2.11 which shows the comparison of stress strain curve between a brittle material and also a ductile material.

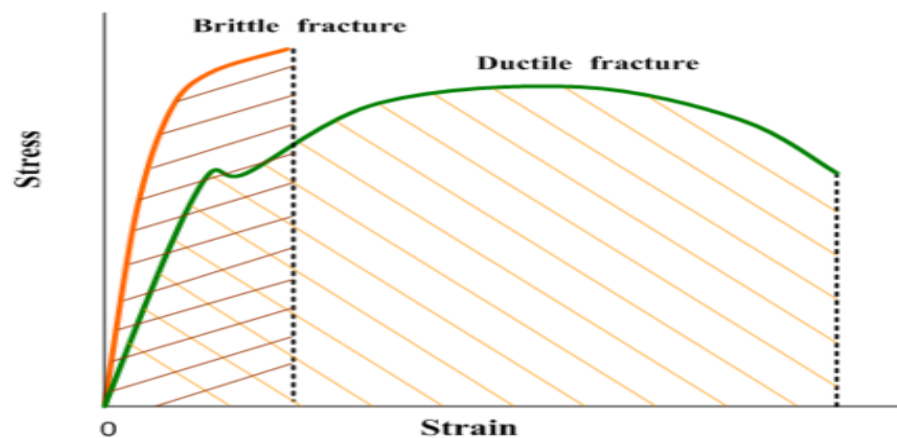


Figure 2.11: Comparison of Stress Strain Curve between a Brittle Material and also a Ductile Material [15]

2.4.3 Ultimate Tensile Strength

The ultimate tensile strength indicates the maximum engineering stress level that a material can withstand without rupture in a tensile test. In figure 2.10, we can see that the ultimate tensile strength is the highest point in the stress strain curve where the line is becoming flat. In ductile material, the strain hardening occurs and the stress will increase until the testing specimen is completely broken down. The strain hardening is occurs due to dislocation movements happened inside the crystal structure of the material. [16]

2.5 POLYMER

Material selection and processing has becoming more and more challenging and important currently. One of the material that becoming more popular is polymer. [19]

The reason is that polymer exhibits properties of good electrical insulation, good corrosion resistance, light in weight and also cheaper in cost if compare with other material such as metallic material. Polymer is the substance that composed of repeating structural units that linked together into a long chain molecule through the process call polymerization. Polymerization is a process that linked all the monomers together into a large molecule with the presence of certain catalyst. Generally, there are four types of polymer linkage, which are branched, network, linear and cross-linked. The different type of linkage can have different effect on the properties of polymer. [18]

The linear polymer is held by strong intermolecular forces of attraction and therefore linear polymer has a high melting point, tensile strength, and also high density. Figure 2.12 below showing schematic illustration of linear polymer.



Figure 2.12: Schematic Illustration of Linear Polymer [20]

Branched polymer will have side branched linear chain attached to the main linear branch, this will change the properties of the polymer because the properties is no longer only depend on the monomer but also the arrangement. [21] The branched polymer exhibits lower density, lower boiling point and lower tensile strength because branch chain polymer are usually irregularly packed. Figure 2.13 below shows schematic diagram of branched polymer.

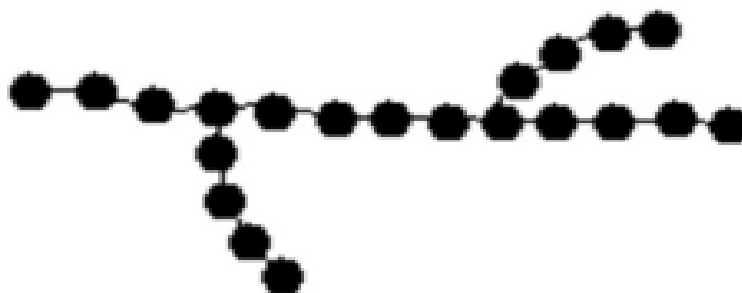


Figure 2.13: Schematic Illustration of Branched Polymer [20]

Cross-linked polymer contains adjacent chains linked by covalent bond. This type of polymer exhibits properties of strength, hardness, brittleness, stiffness, and better dimensional stability. Figure 2.14 and 2.15 shows the schematic diagram of the cross-linked polymer and network polymers respectively. A network polymer is a highly cross-linked polymer.

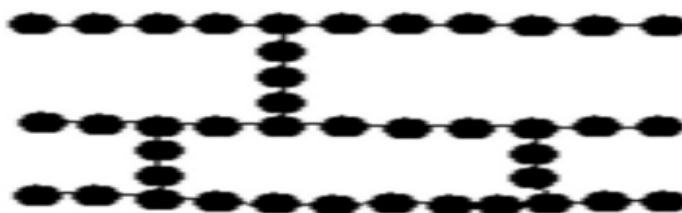


Figure 2.14: Schematic Illustration of Cross-linked Polymer [20]



Figure 2.15: Schematic Illustration of Network Polymer [20]

In the thesis of “Low to High Strain Rate Deformation of Amorphous Polymers”, two amorphous polymers, polycarbonate and polymethyl methacrylate has undergone an experimental and analytical investigation to find out the mechanical behaviour of them. The testing strain rates are range from 10^{-4} to 10^4 s^{-1} . This spectrum of strain rates was achieved in tension tests using a dynamic mechanical analyzer (DMA), a “servo-hydraulic testing” machine and a split-Hopkinson pressure bar. Tension tests on the DMA were used to characterize the time-temperature dependence of the elastic storage moduli and the loss factors of the two polymers. DMA testing provided a quantification of the rate-dependent shift of material transitions, and led to an analytical description of the elastic modulus at all rates and temperatures. Uniaxial compression tests were performed at strain rates of 10^{-4} s^{-1} to 1 s^{-1} using a servo-hydraulic testing machine. Stress strain behavior of the materials was found to be rate-dependent in this regime, with yield strength increasing in a linear and non-linear fashion for PC and PMMA , respectively, with the logarithm of strain rate . High strain rate tests were performed using an aluminum split-Hopkinson pressure bar. Material strengthening as a function of strain rate was observed to be dramatically greater in this high-rate regime for both materials . A physically-based constitutive model for large strain deformation of thermoplastics, which has been shown to be

accurate at quasi-static rates of deformation, was then extended to encompass very high rate conditions. The revised model is shown to be predictive of the behavior of these materials over a wide spectrum of strain rates. This model uses the viscoelastic material transitions quantified in DMA testing, as well an understanding of the molecular mechanisms of resistance to deformation at high rates, to capture the overall transition of the stress-strain behavior to a regime of increased rate-dependence of yield.

There are two main industrially important polymeric materials, which are plastics and also elastomers. The polymer plastics can be further divided into two classes, thermosetting plastics and thermoplastics. [17] They are being differentiated by how they are chemically bonded and structurally bonded. The type of polymer that we focus in this project is thermoplastics.

2.5.1 ELASTOMER

The elastomer is polymer with the ability to deform at large deformation and then elastically return back to their original state. This is because elastomer consists of cross-linked polymer, and they supply a lot of intermolecular forces of attraction to help the polymeric chain to return back to its original state. Figure 2.16 below shows the elastomer in unstressed state. We can clearly see from the figure, the polymer is amorphous and consists of cross-linked polymeric chain.



Figure 2.16: Elastomer in Unstressed State [22]

When there is a stress applied at the cross-linking, the chains will straighten themselves and this process may involve partial uncoiling and also untwisting of corresponding chains. The effect will be shown at the figure 2.17 below. After the applied stress is removed, the elastomeric chain will return to their original state.

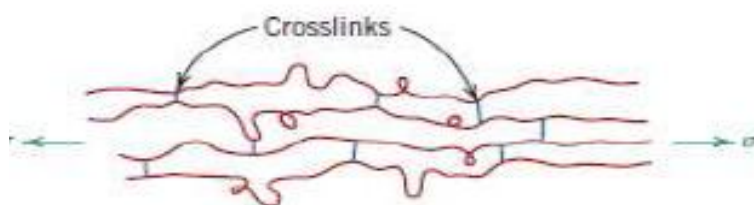


Figure 2.17: Elastomer in Stressed State [22]

2.5.2 THERMOSETTING POLYMER

Thermosetting polymer is permanently set themselves after the thermal application. Thermosetting plastics or in other words thermosets are network polymer which has a permanent shape. They are also high thermal stability, high rigidity and good resistance to creep deformation. There is a lot of covalent cross linking between

adjacent molecular chains bond, they will bond together and form a hard infusible and insoluble product upon heating. This means that it does not soften easily on heating and make them a good choice when we need to use them in high temperature condition. Besides that, they are also stronger than thermoplastics polymers and also more brittle than the thermoplastics polymers. Figure 2.18 below shows the schematic molecular structure of the thermosetting polymer.



Figure 2.18: Schematic Molecular Structure of the Thermosetting Polymer [23]

2.5.3 THERMOPLASTIC POLYMER

The thermoplastic polymer is a type of polymer that can be softened when the temperature is rising and can be hardened when the temperature is dropping. [24] There is no cross-linking in the molecular chain of thermoplastic polymer and this is clearly shown in the figure 2.19. If there is a rise in temperature, the Van Der Waal forces will begin to get diminished and therefore no effect will occur when the stress is applied. This will cause relative movement in the adjacent chains of polymer, when the chains move more and more freely, this is the time the softening will occur. This type of polymer can be easily produced by melting them through heating and then molding

them into the desired shape and cooled it to get the final product. The common example of thermoplastic included polystyrene, polyethylene, and polypropylene. This project will focus only on the polypropylene, to determine the tensile deformation properties of polypropylene.

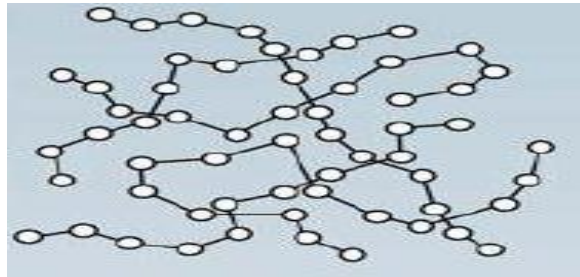


Figure 2.19: Schematic Molecular Structure of the Thermoplastic Polymer [23]

2.6 SUMMARY

As a summary, the thermoplastic polymer is a type of polymer that can be easily molded into the desired shape as they can be softened on heating and hardened on cooling. Polypropylene is one of the thermoplastics; it has excellent chemical and heat resistance and low density. Polymer has been used in many applications such as food packaging and car components as well. The sheet material made by polypropylene will be shaped into a dog bone shape. The dog bone shaped testing specimen will undergo uniaxial stress strain analysis. The simulation result will be produced by using ANSYS APDL to perform uniaxial stress strain analysis. ANSYS APDL is a tool commonly used to perform analysis on the material to determine how the material reacts in the real world situation. Lastly, the simulation result will be interpreted to find out the service behavior of the material.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter will discuss about the methodology that is going to use in this project. This chapter will provide some information and some basic overview of the investigate focus, research methodology about this project. Besides that, a flow chart to show the process flow of the project will also be included in this chapter. Lastly, this chapter will also discuss about the material software that will going to use in this project.

3.2 FLOW CHART

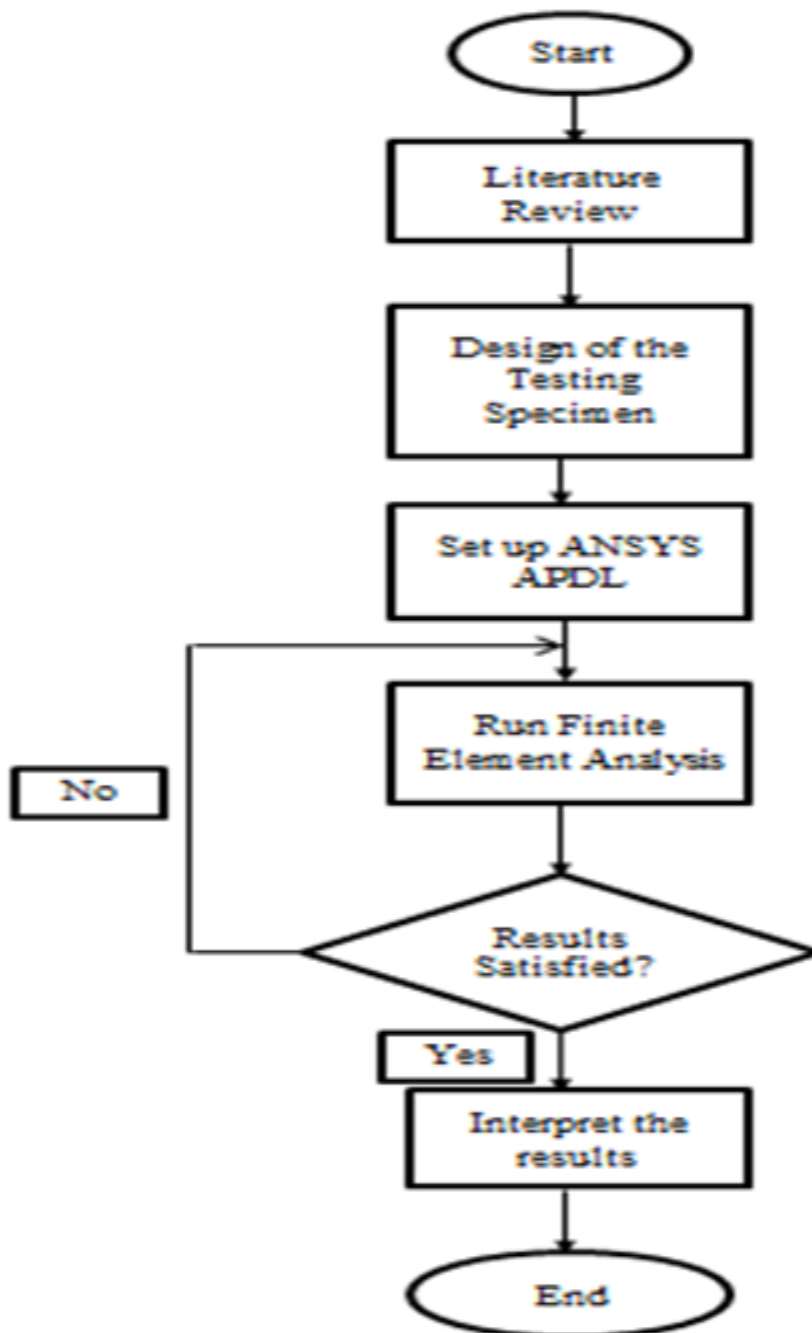


Figure 3.1: Flow Chart

3.2.1 Project Description

The material for testing specimen will be polypropylene. ANSYS APDL is the software to be used in this project. First, a model will be shaped into dog bone shape and the dimension will follow standard ASTM D638 in the simulation software. The simulation results can be gotten with the help of ANSYS APDL. ANSYS APDL is a useful tool to perform finite element analysis. After the ANSYS APDL is setting up, then the finite element analysis will be performed. The analysis is uniaxial stress strain analysis and the results will be recorded down. After the simulation result is obtained, the simulation results will be the ideally results. Lastly, there will be some discussions on the final results and a conclusion will be made for this project.

3.3 MATERIAL AND SOFTWARE

The software needed for this project is ANSYS APDL. In this section, we will discuss and provide the information for all these hardware and software needed.

3.3.1 Dog Bone Shape Testing Specimen

Dog bone shape testing specimen is commonly used in tensile testing. Its design consists of a gauge section in the middle and also two shoulders at both ends each, the shoulder is wider in size than the middle gauge section. Has the purpose to make sure that when it is loaded with the tensile force, the stress concentration only happens in the middle gauge section. In this project, the dimension of the dog bone shape will be

following the standard of ASTM D638. Figure 3.2 below will show the dimension of the dog bone shape testing specimen.

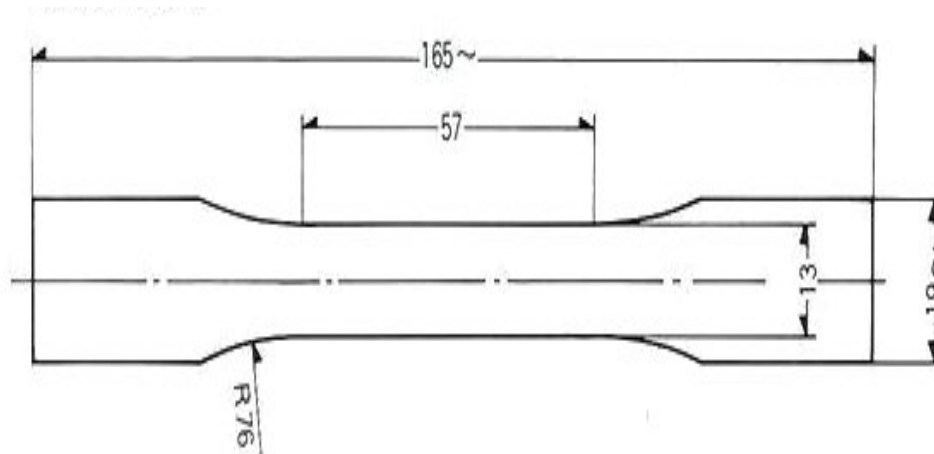


Figure 3.2: Dimension of Dog Bone Shape Testing Specimen

The material that is going to use to cut into testing specimen is polypropylene. The common advantages of polypropylene included excellent chemical resistance, able to be recycled, excellent heat resistance, easy to manufacture and low density. Polymer is well suited for many different types of uses, included flexible and rigid packaging, automotive interior and exterior, and also consumer products. Polypropylene is recyclable material, it is an important consideration in producing product. Furthermore, it can be incinerated without any toxic emissions.

3.3.2 ANSYS Parametric Design Language

ANSYS Parametric Design Language (APDL) is software to allow users to parametrically build a finite element model, solve it and obtain the results. It can also

extract the characteristic results parameter such as maximum stress. Furthermore, ANSYS APDL provides the possibility to use input variable as well as result parameter in arithmetic expression. [4] In this project, ANSYS Parametric Design Language (APDL) is used to perform uniaxial stress strain analysis to determine the mechanical behavior of the polypropylene. The final result of this testing will be a stress strain curve. From there, we can know the mechanical behavior of the polypropylene. Figure 3.3 below shows the logo of the ANSYS.

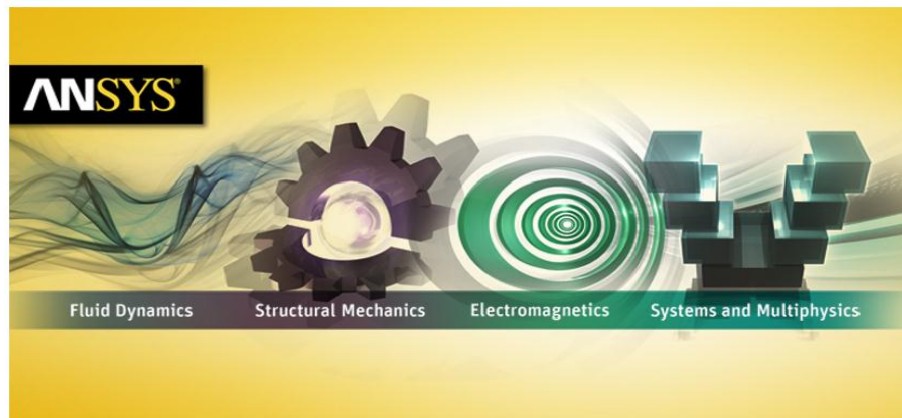


Figure 3.3: Logo of ANSYS

3.4 PROCEDURES TO SET UP ANSYS

In this section, the step by step procedure to set up the ANSYS to perform the finite element analysis will be provided. First of all, the shape of the dog bone is drawn. The dimensions will follow the standard of ASTM D638.

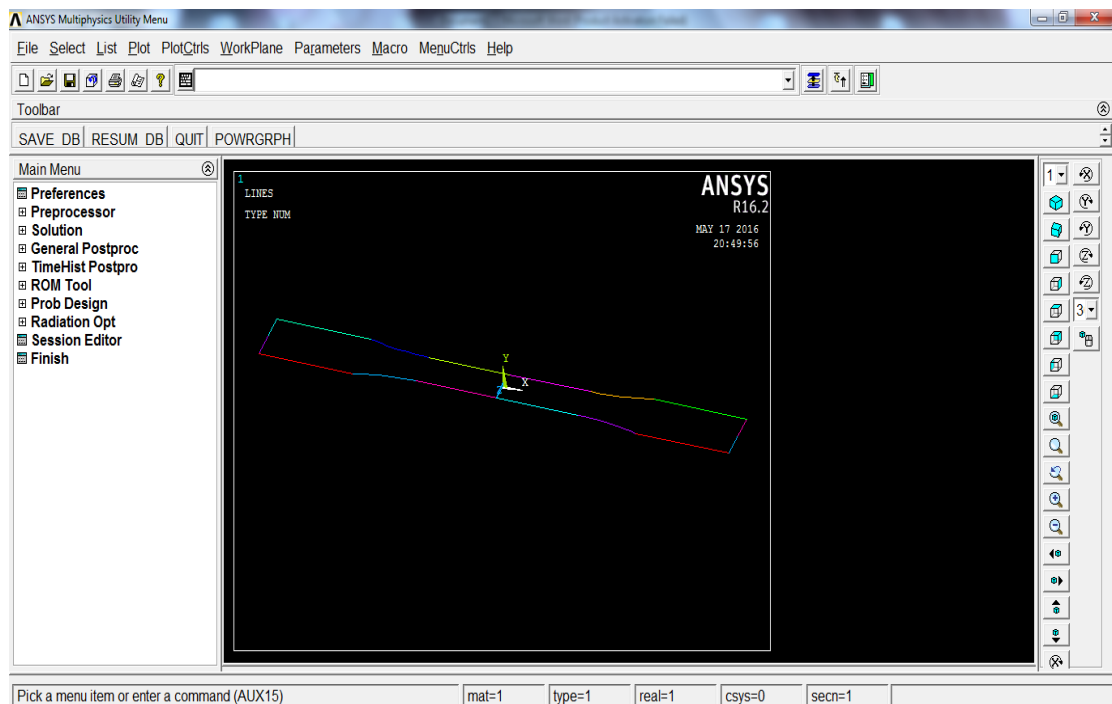


Figure 3.4: Drawing of Dog Bone Shaped Specimen

In the figure 3.4 above shows the drawing of dog bone shaped specimen. The overall length of the testing specimen is 165mm, the width of grip section is 19mm, the gauge length is 57mm and the sample dimension of the testing specimen according to ASTM D638 is shown at figure 3.5 below.

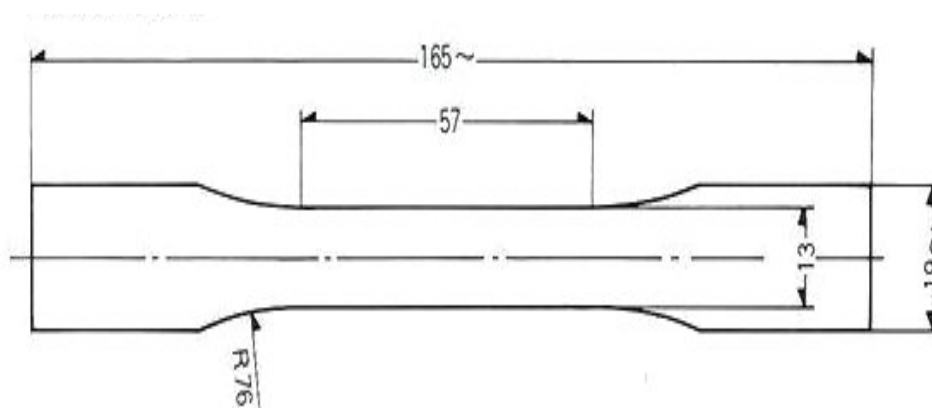


Figure 3.5: Dimension of Dog Bone Shape Testing Specimen

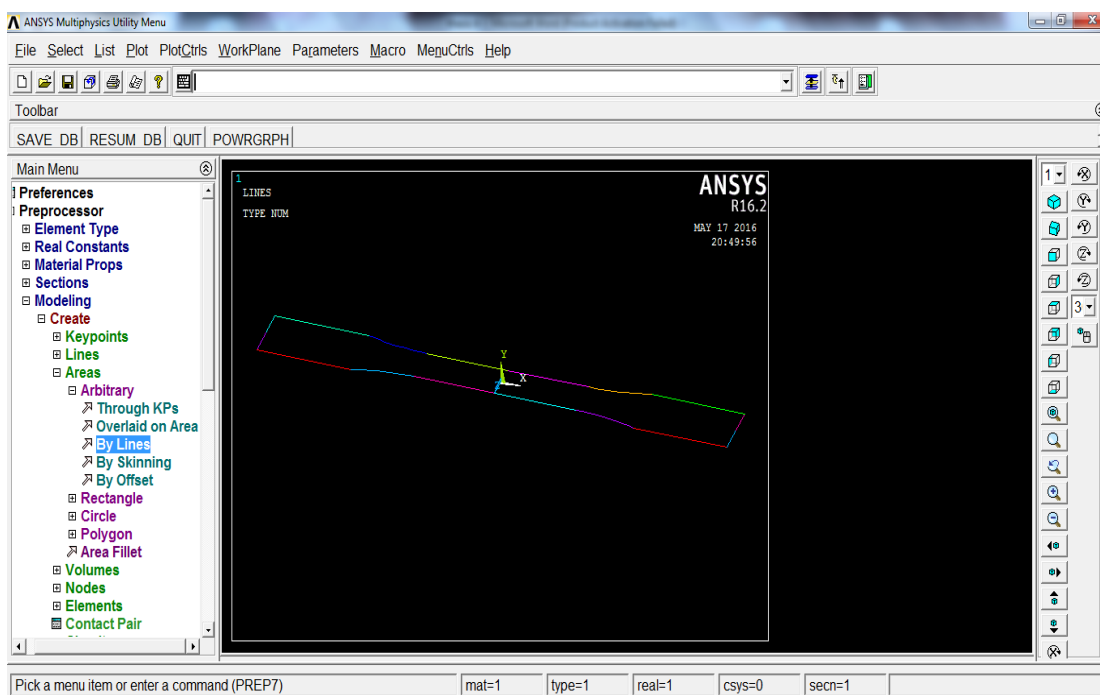


Figure 3.6: Creation of Area

In the figure 3.6 above shows the step to create the area. An area is needed to be created by line under the modeling section.

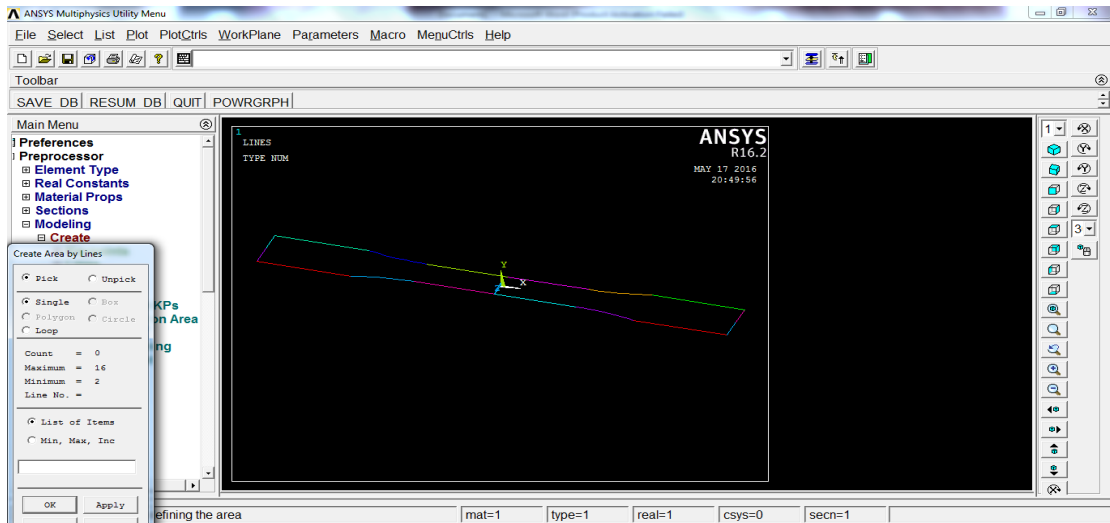


Figure 3.7: Small Window of Create Area by Lines

In the figure 3.7 above shows the small window of create area by lines. After the “by line” button is clicked, it will pop out a small window at left side.

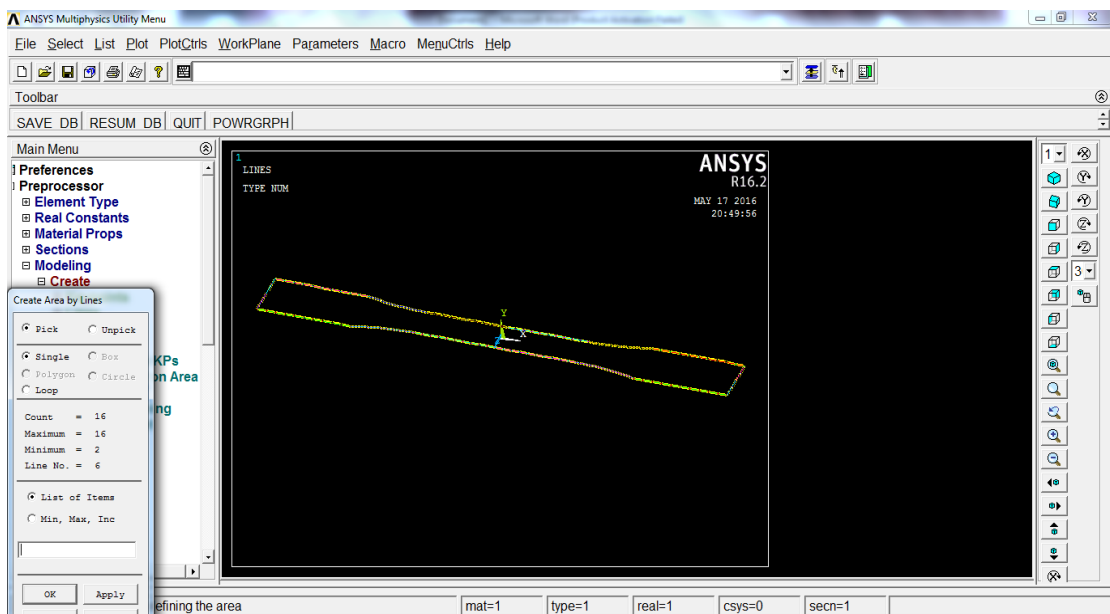


Figure 3.8: Selection of the Lines

In the figure 3.8 above shows the line needed is chosen to create the area. Then, the “apply” button is clicked to create the area and then clicking “ok” button to confirm it. The area outcome will look like in the figure 3.9 below.

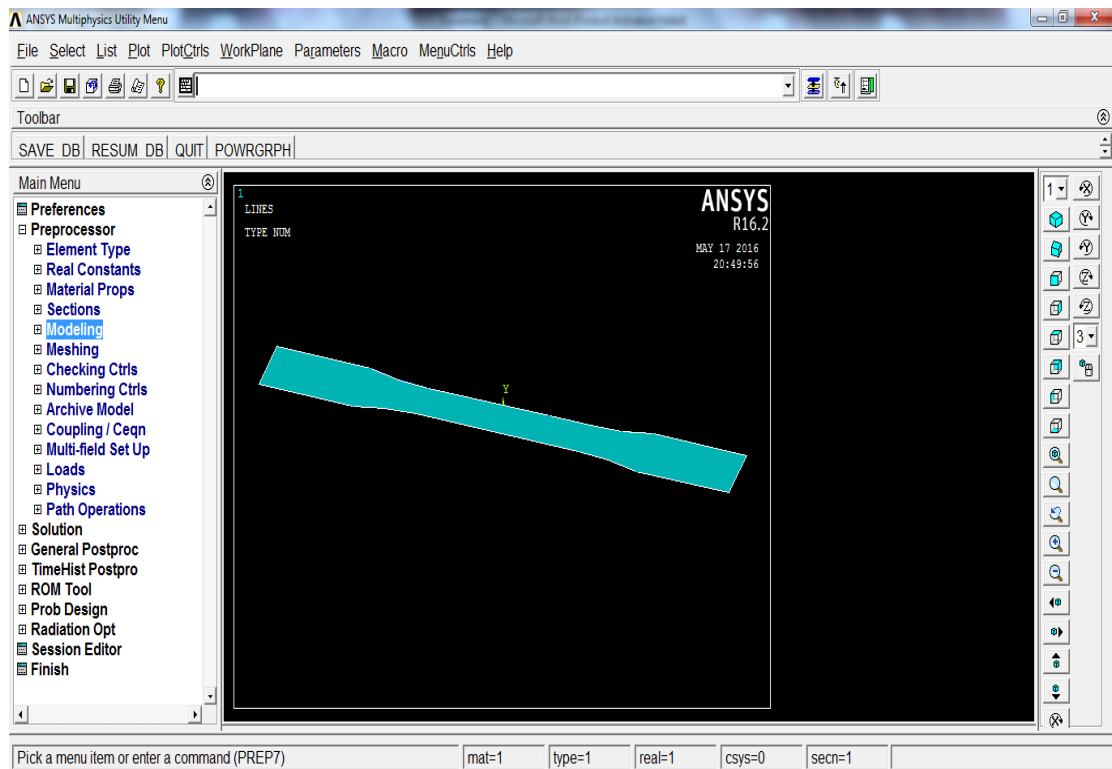


Figure 3.9: Area of the Model

After that, the element type will be added to the model. The step to complete that is by clicking the “add/edit/delete” button under the element type. A window of element type will pop out after that and is shown in figure 3.10 below.

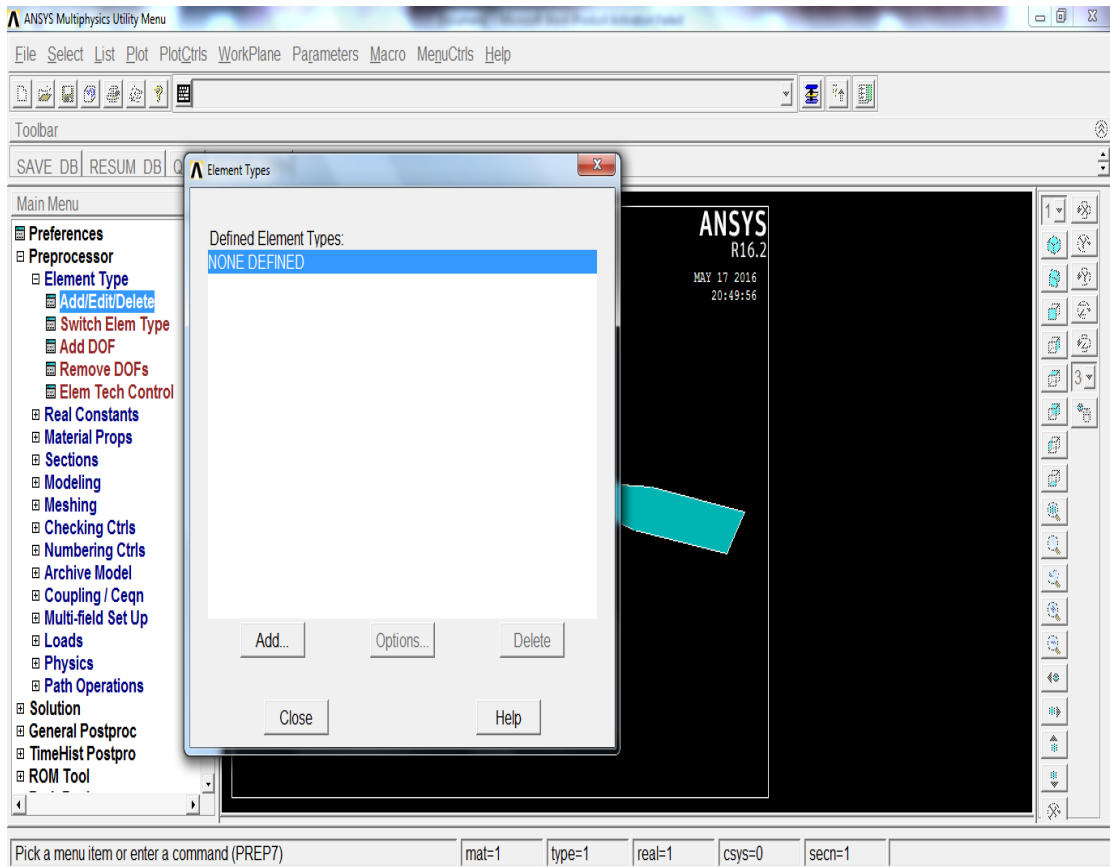


Figure 3.10: Selection of Element Type

The element type chosen here is shell 63 and is shown in figure 3.11 below. According to the thesis, it shows that shell element is recommended when the thickness of the model is very thin. The shell 63 chosen because the stress stiffening and large deflection capabilities are included.

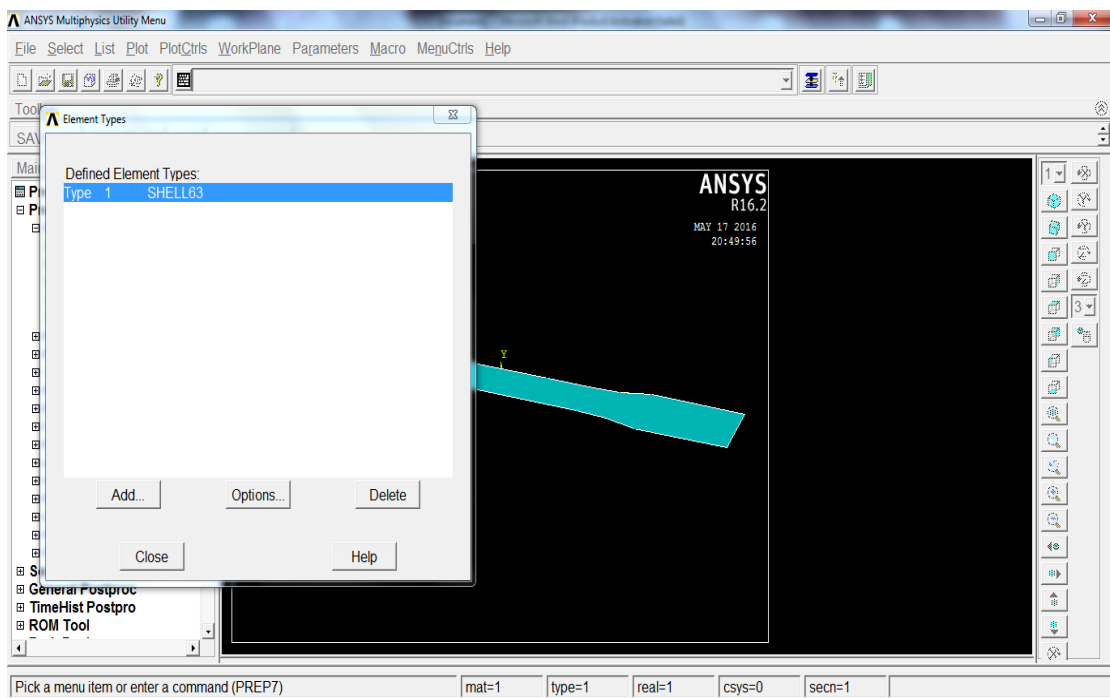


Figure 3.11: Element Type of Shell 63

Next, the real constant is added. To do this, the “add/edit/delete” button under the real constant is clicked and a window of real constant will pop out later and is shown in the figure 3.12 below.

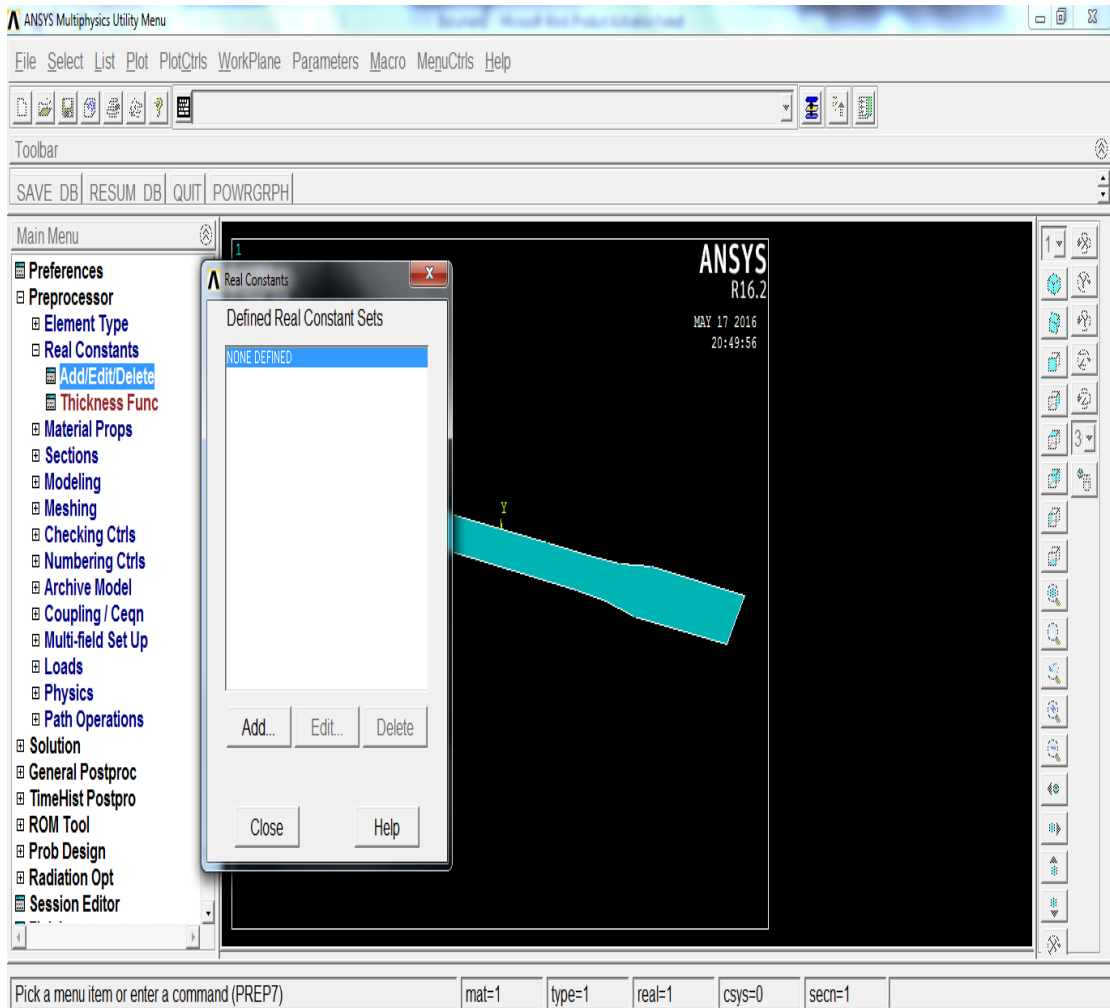


Figure 3.12: Defined of Real Constant

After the “add” button is clicked, another window of element type for real constant will pop out. From here, the “ok” button is clicked to choose the type shell 63 element. The detail is shown in the figure 3.13 below.

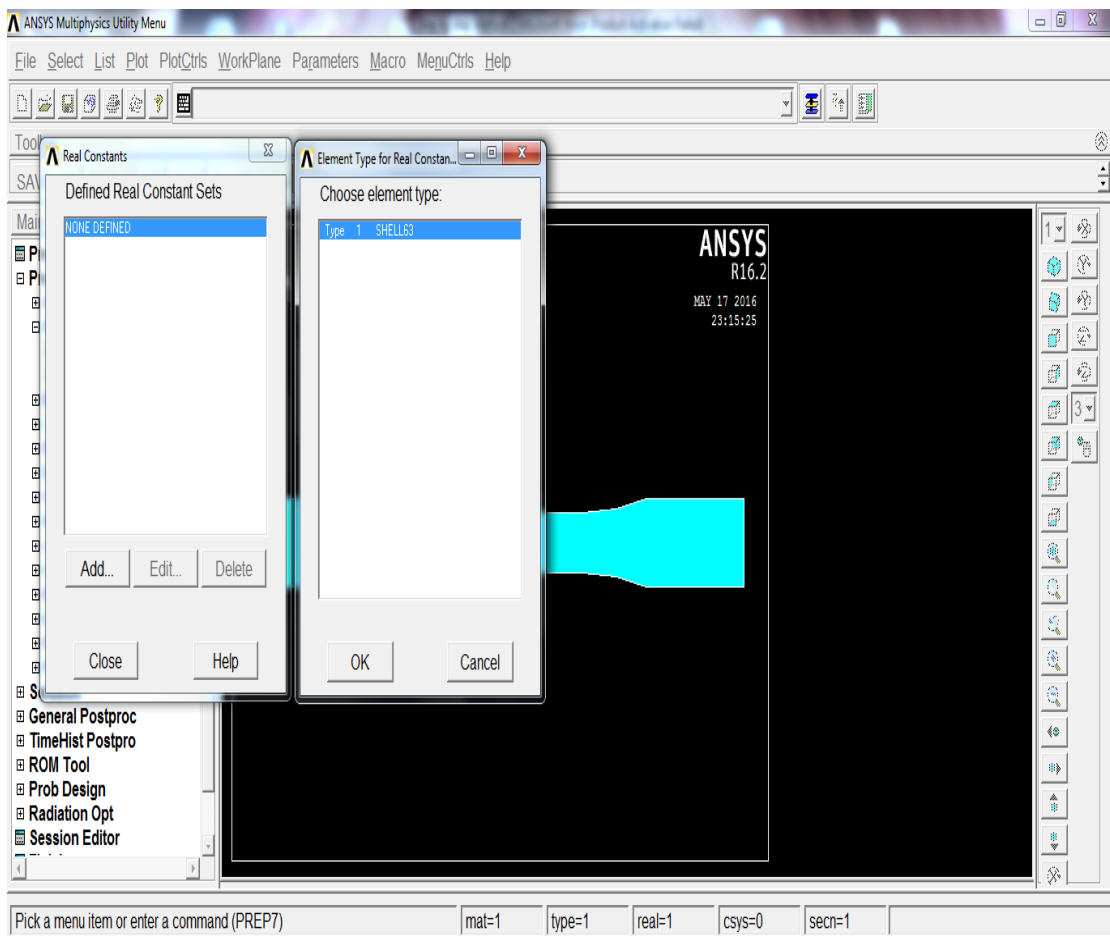


Figure 3.13: Window of Element Type for Real Constant

After that, the thickness is needed to be set by setting the TH(I) to 0.001. If the element has a constant thickness, only TK(I) need be input. Then, clicked the “apply” button and “ok” button to confirm it. The detail will be shown in the figure 3.14 below.

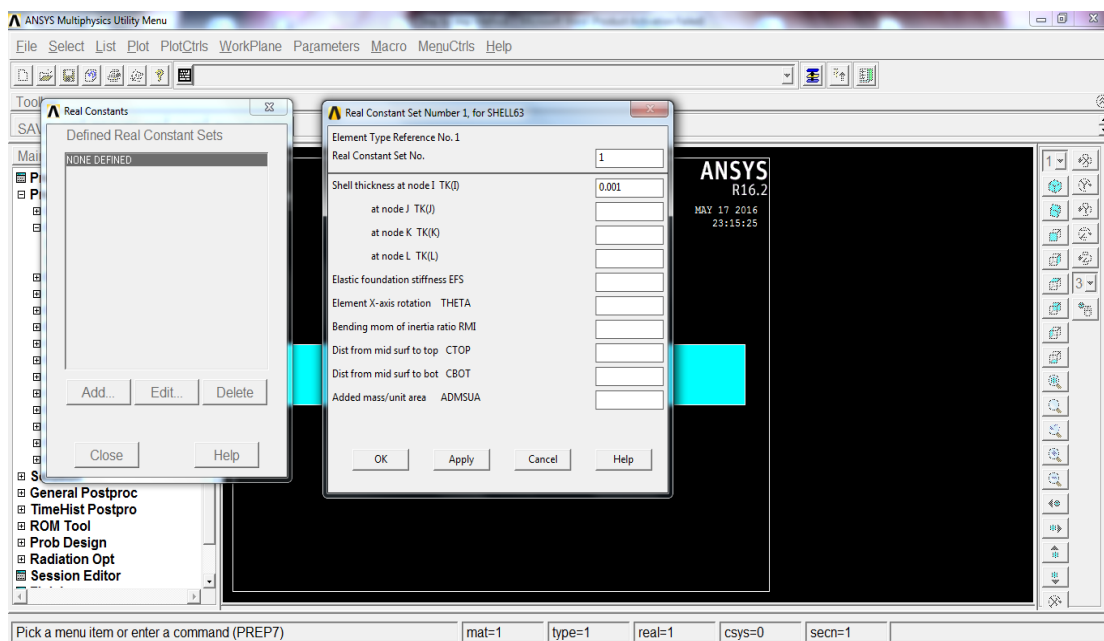


Figure 3.14: Set Up of Thickness

Next step, the material models will be set under the material properties.

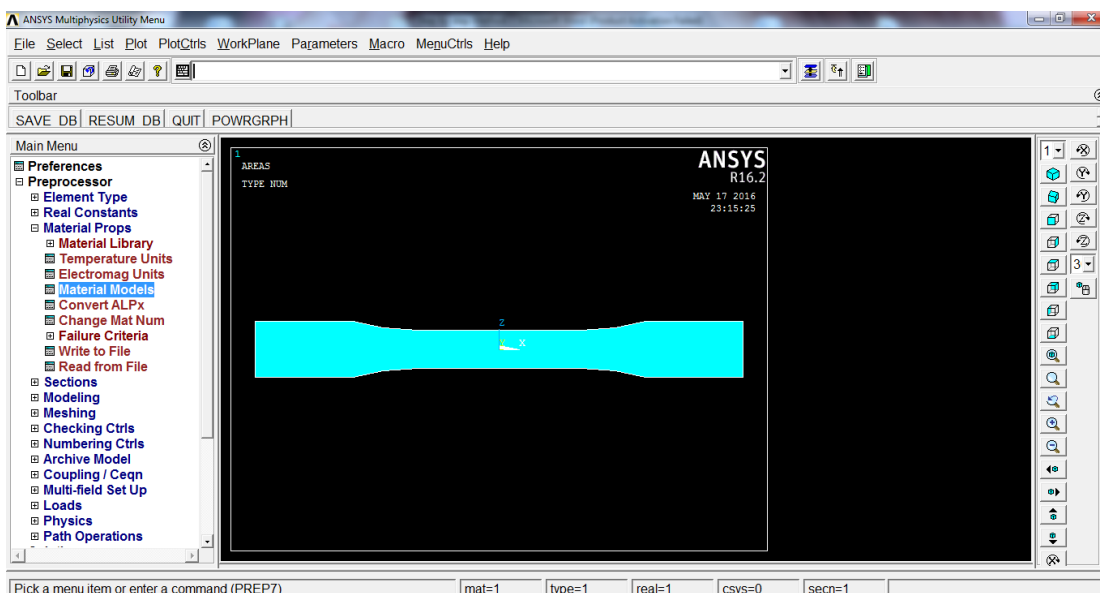


Figure 3.15: Set Up of Material Properties

After that, the young's modulus and poison ratio of the material is set. Then, mesh tool is opened under the meshing.

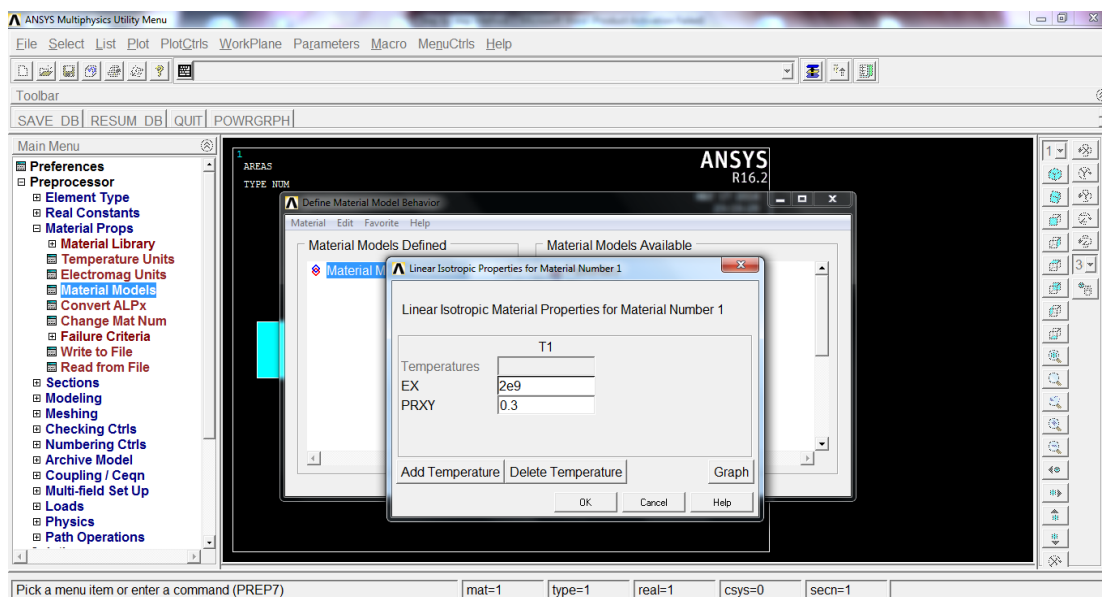


Figure 3.16: Set Up of Young's Modulus and Poison Ratio

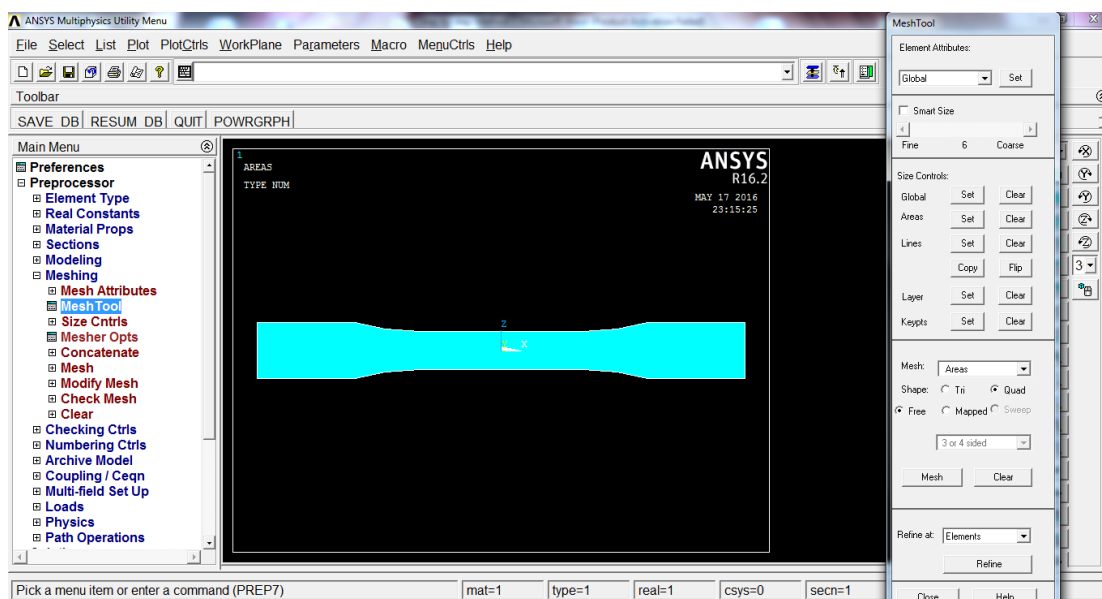


Figure 3.17: Mesh Tool

In the mesh tool, the meshing attributes are opened by clicking the “set”. In the meshing attributes, the element type must be confirmed to be shell 63. After checked, clicking “ok” button to confirm it.

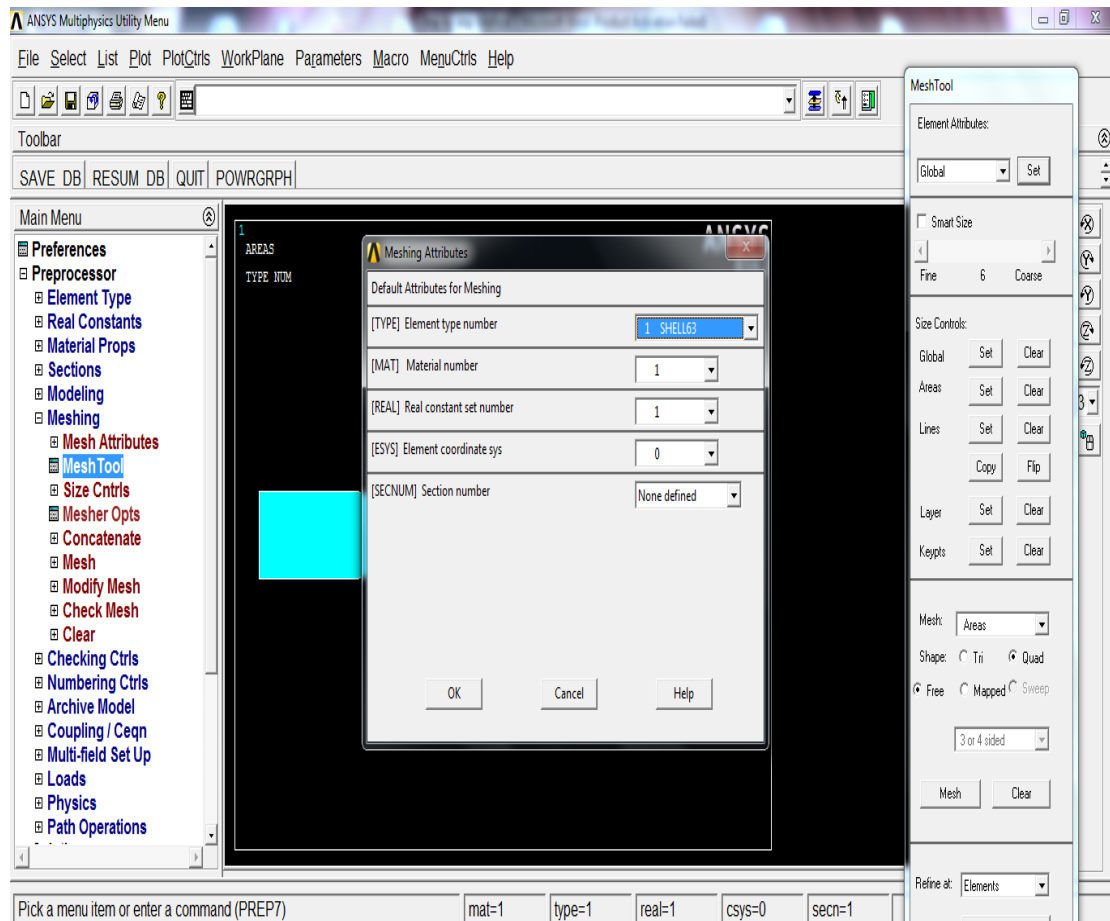


Figure 3.18: Checking of Meshing Attributes

Next, under the mesh tool, the “set” under the line is clicked. A small window will pop out, and then the line is selected for further action. At here, the line on both sides are selected and then clicked the “ok”.

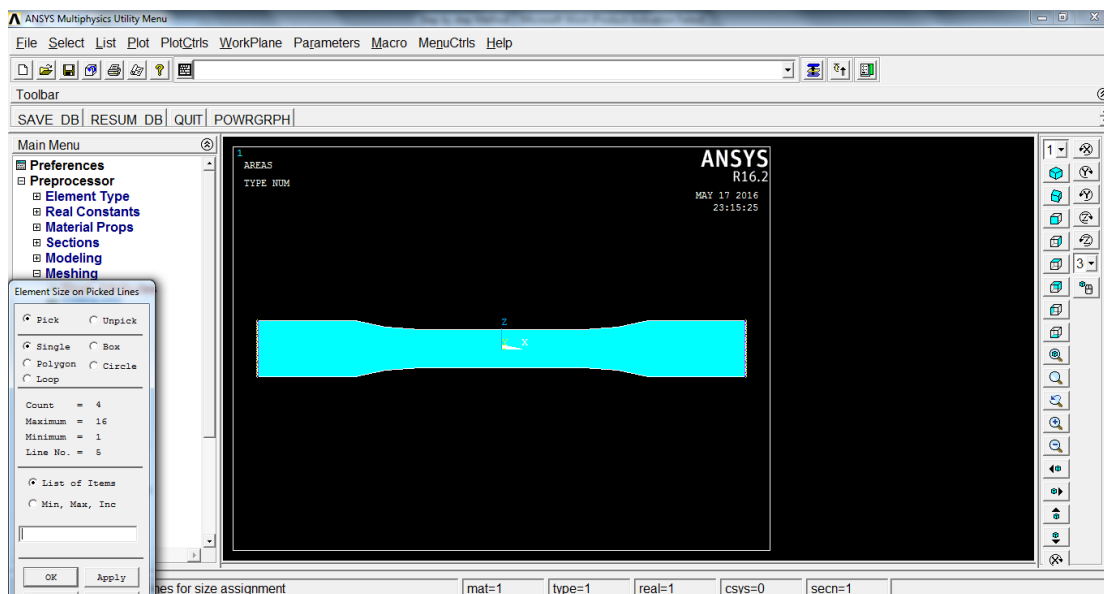


Figure 3.19: Selection of Both Side Line

After the “ok” is clicked, a window of element size on picked line will pop out. At here, the no of element division is set to 25 and click “ok”.

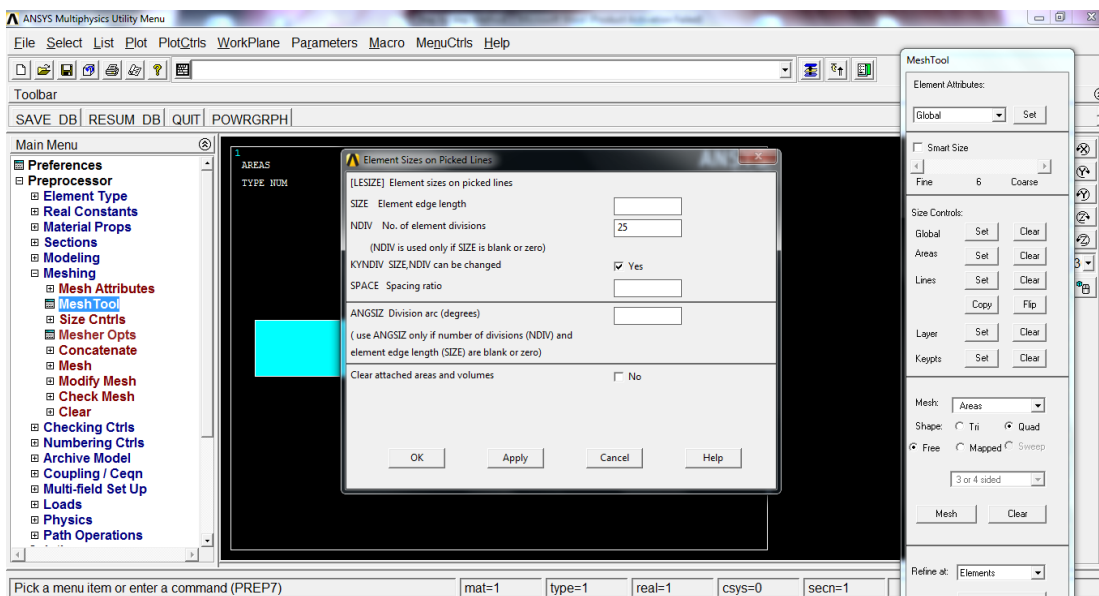


Figure 3.20: Setting of Element Size on Both Side

Then, the same procedure will be applied on the line of the grip section. For the grip section, the no of element division is set to 30.

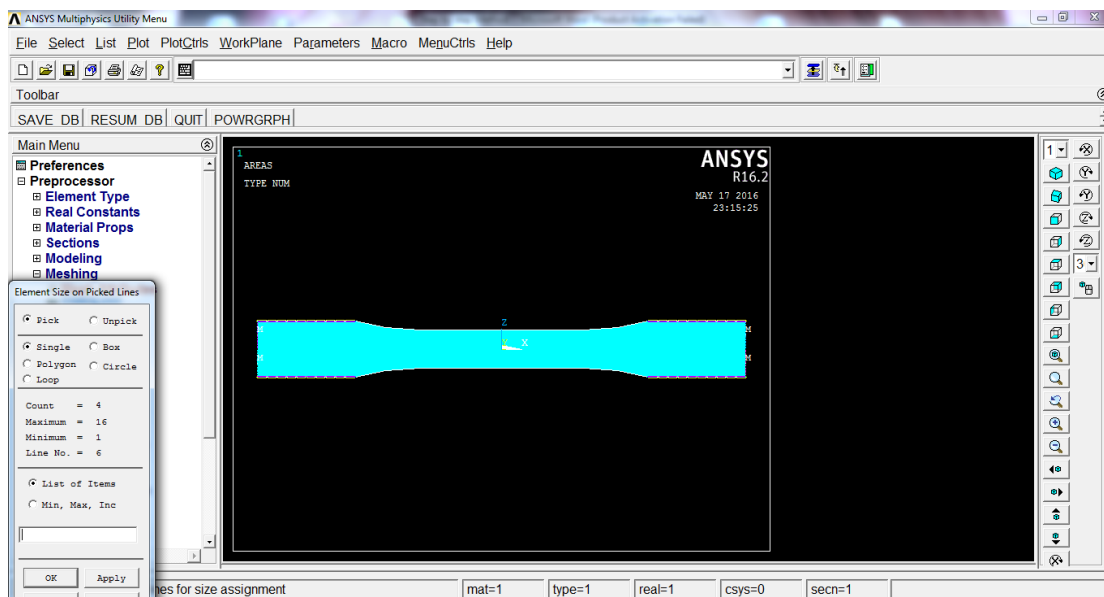


Figure 3.21: Selection of Line at the Grip Section

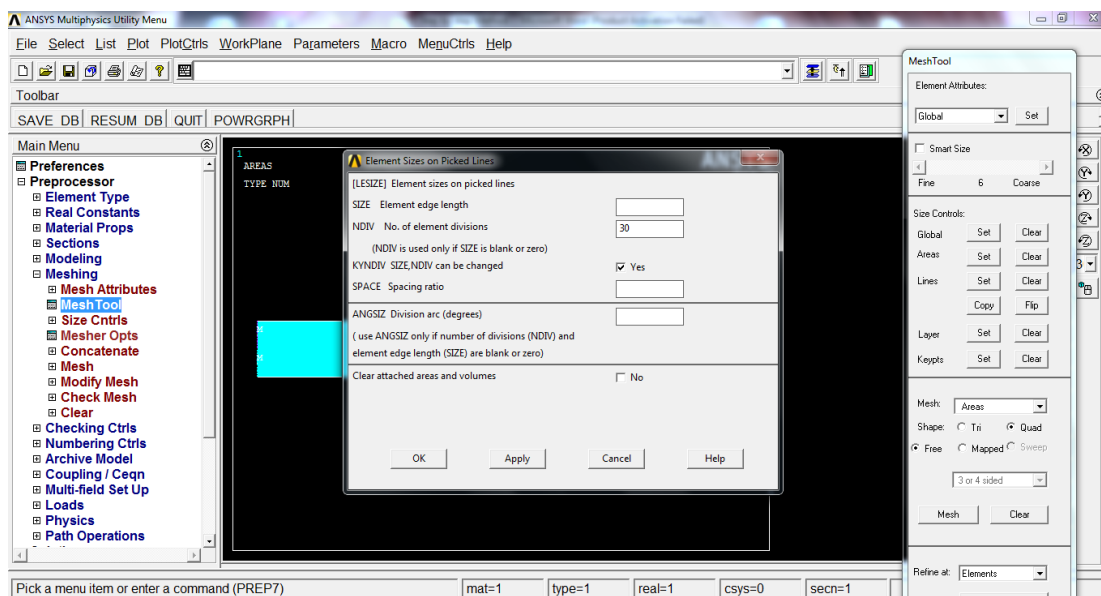


Figure 3.22: Setting of Element Size on Grip Section

Then, repeat the same procedure to the curve line. For the curve line, the no of element division is set to 20.

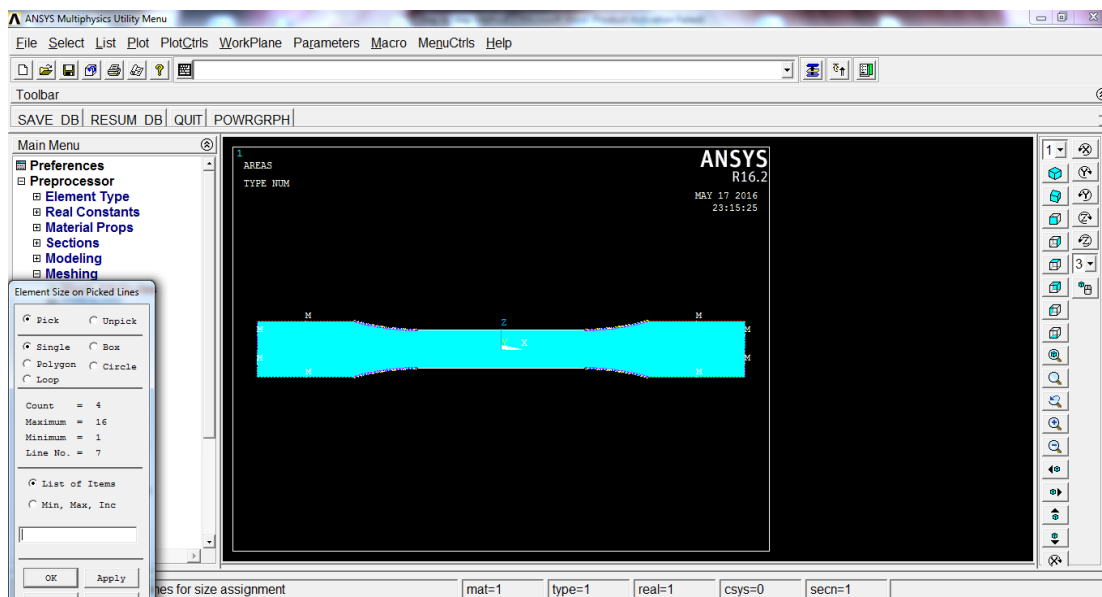


Figure 3.23: Selection of Curve Line

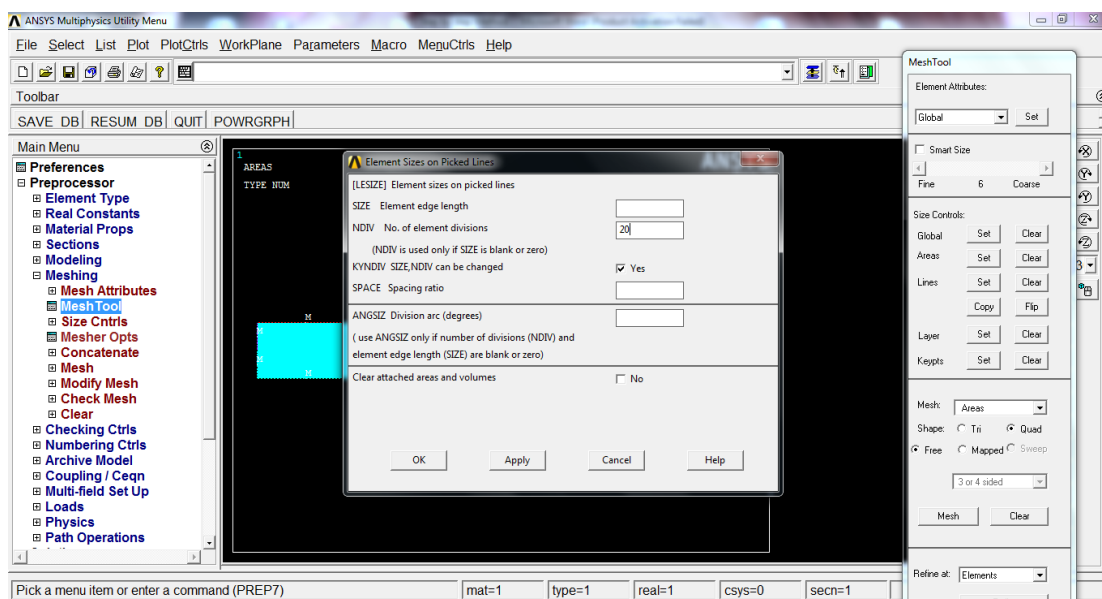


Figure 3.24: Setting of Element Size on Curve Line

Then, the same procedure is repeated for the middle line. For the middle part, the no of element division is set to 75.

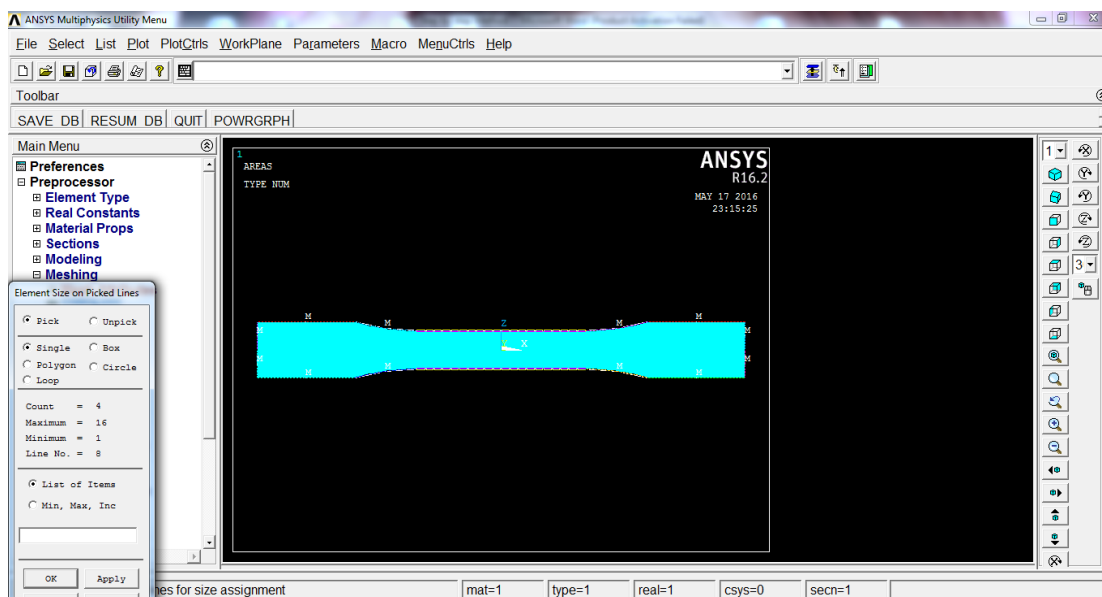


Figure 3.25: Selection of Middle Line

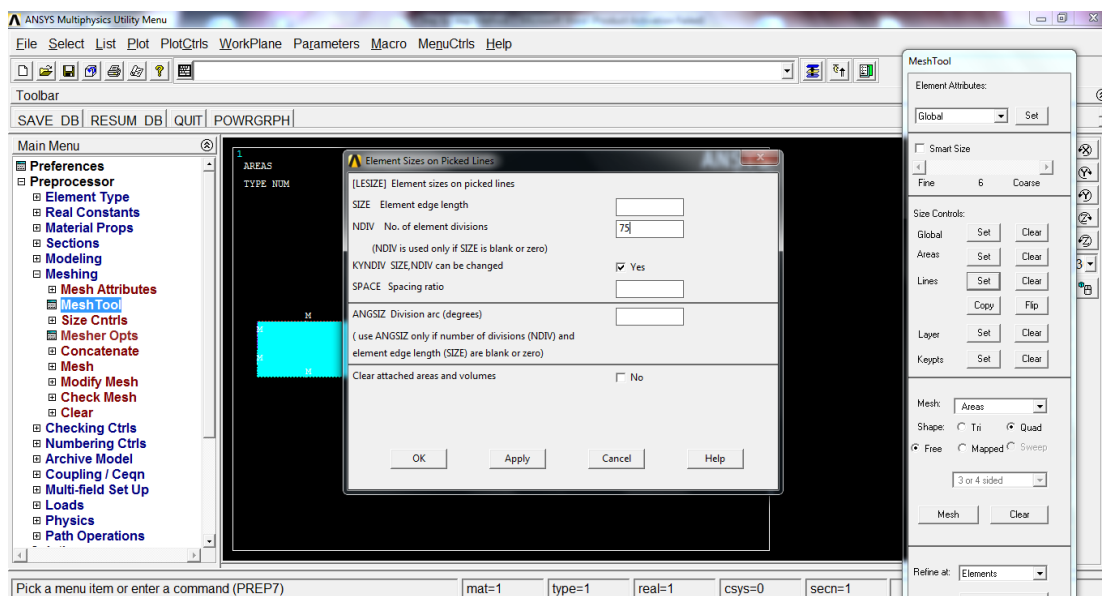


Figure 3.26: Setting of Element Size on Middle Line

Then, under the mesh tool, the “mesh” button is clicked and a small window of mesh area will pop out. At here, the area that is needed to be mesh is selected and clicked the “ok” to confirm it.

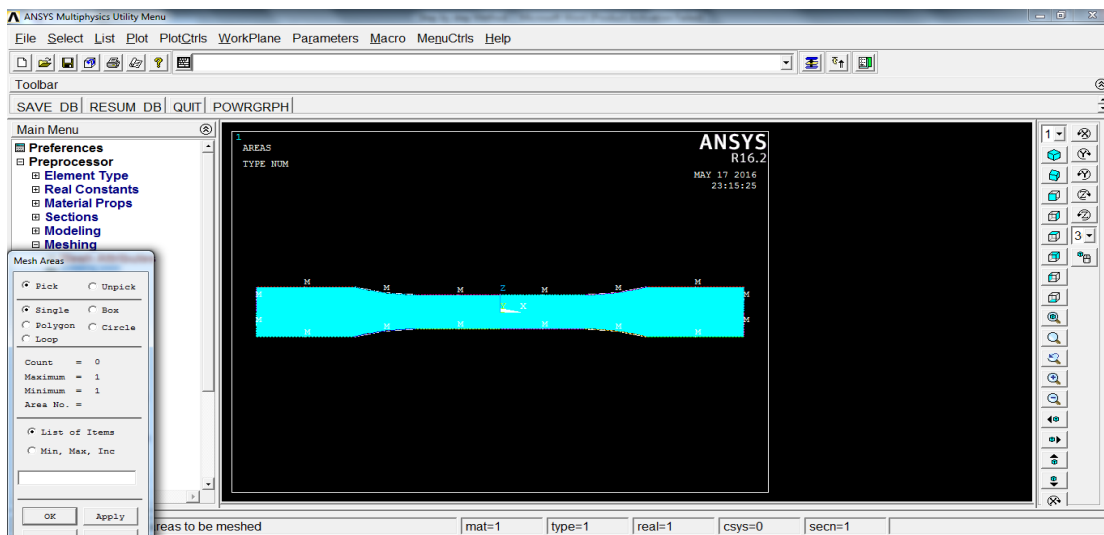


Figure 3.27: Window of Mesh Area

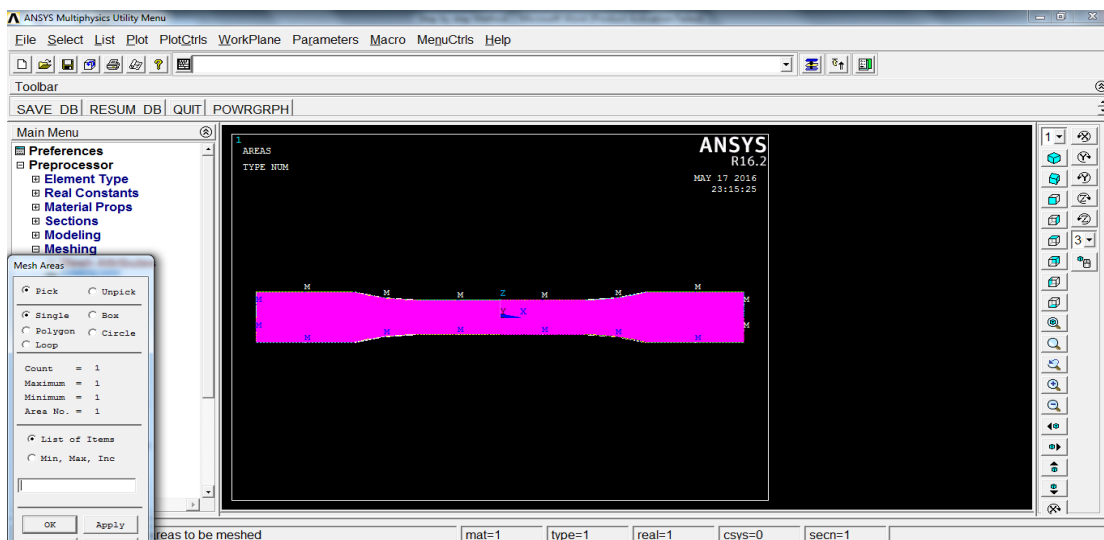


Figure 3.28: Selection of Area

After that, the meshing process will be running. The result is looked lie the figure 3.28 below.

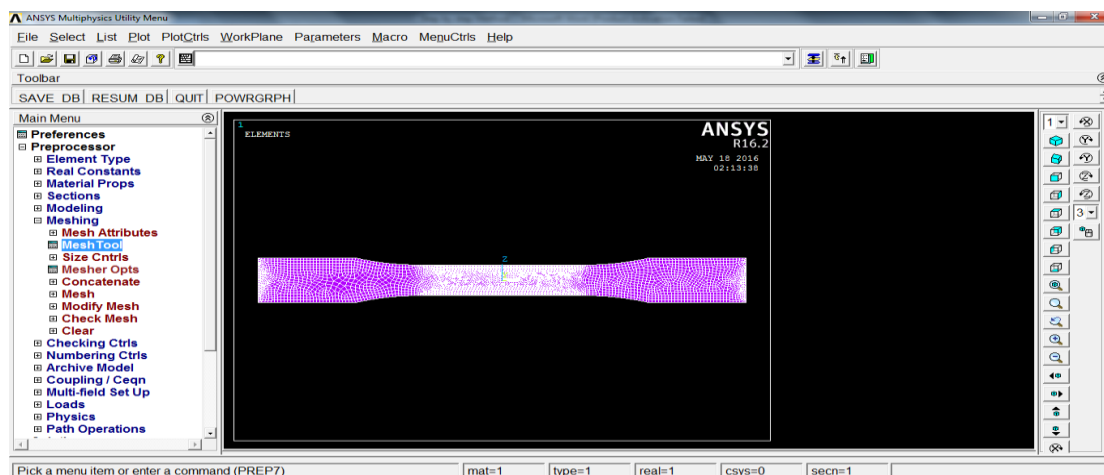


Figure 3.29: Result of Meshing Process

After that, the “structural” is clicked under define loads.

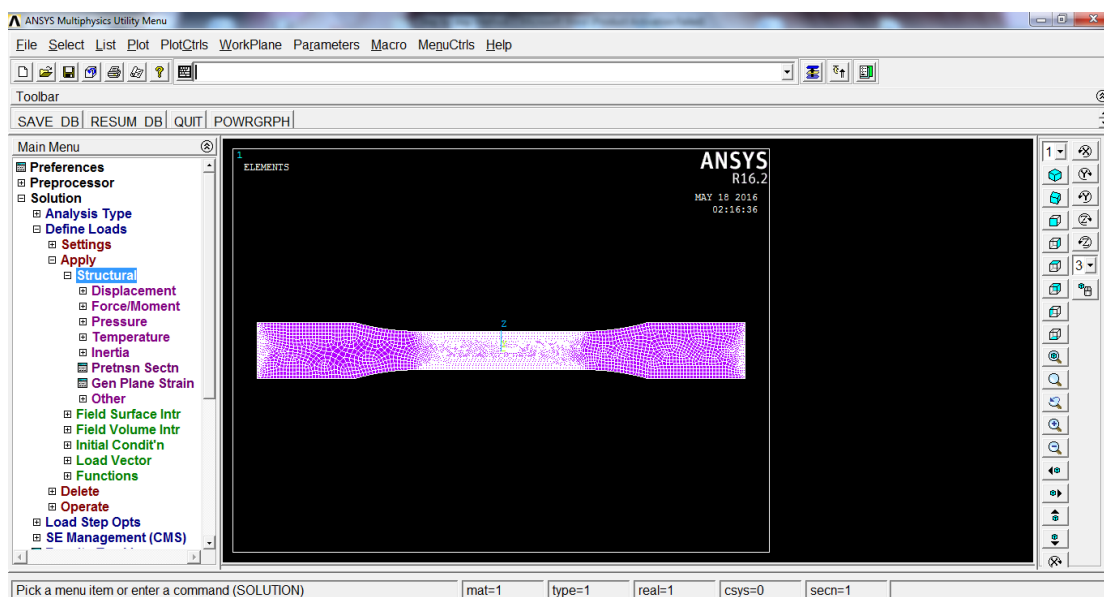


Figure 3.30: Applying Structural Load

From the structural, the “displacement” is clicked and a small window will pop out after clicking the “on node”. At here, the box is ticked and choose the part that is needed to be applied. The “ok” is clicked to confirm it.

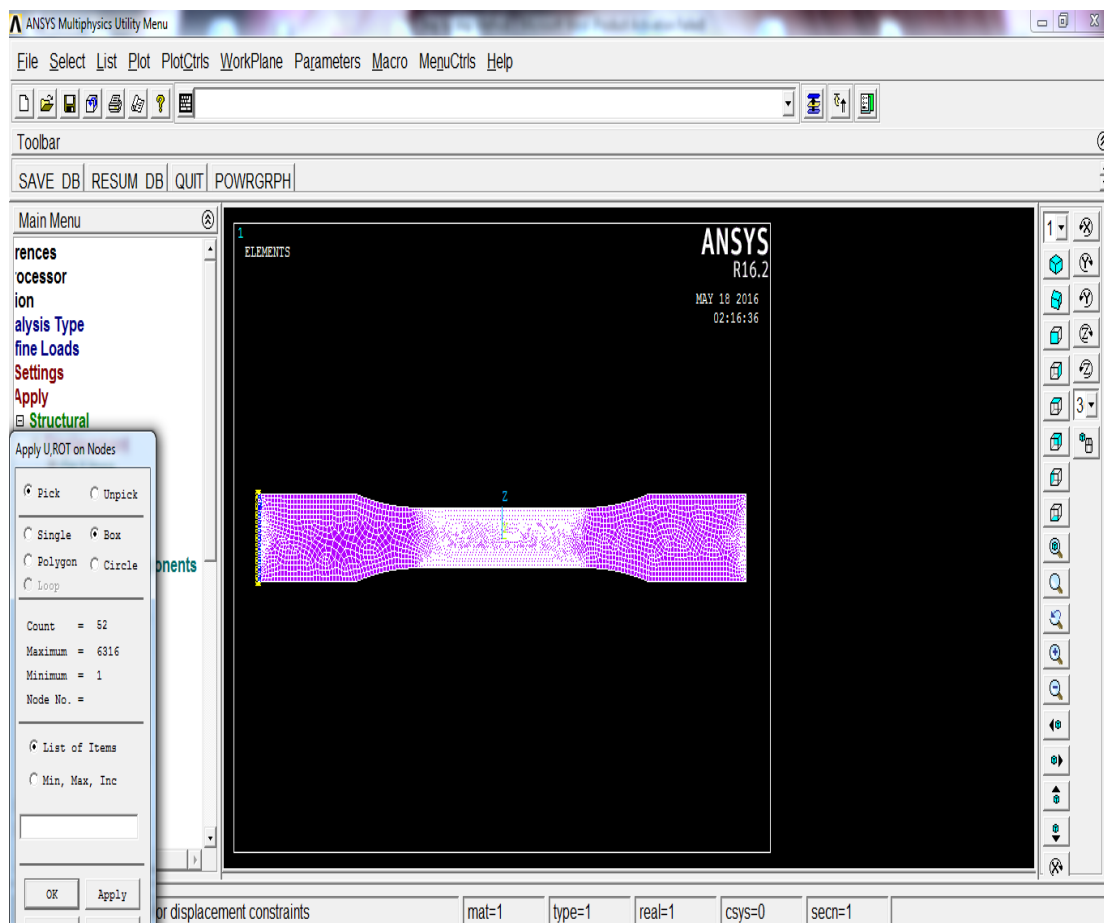


Figure 3.31: Defined of Displacement Load

After that, a window will pop out. At here, the degree of freedom to be constraint is needed to set, and all degrees of freedom are chosen and clicked “OK” to confirm it.

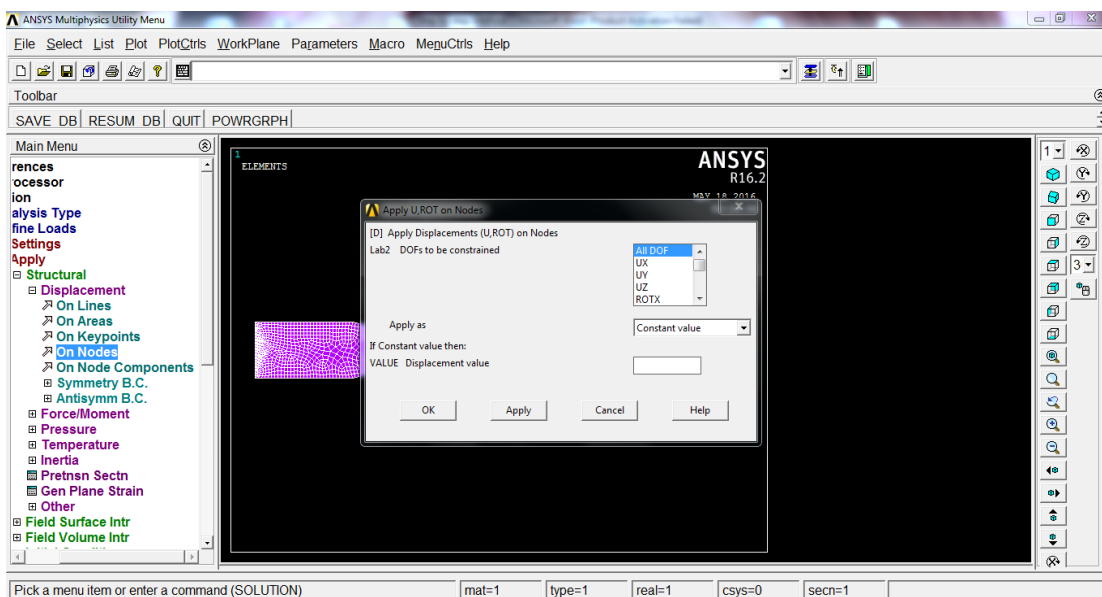


Figure 3.32: Defined of Degree of Freedom

After confirmed, the result will be looked in the figure 3.32 below.

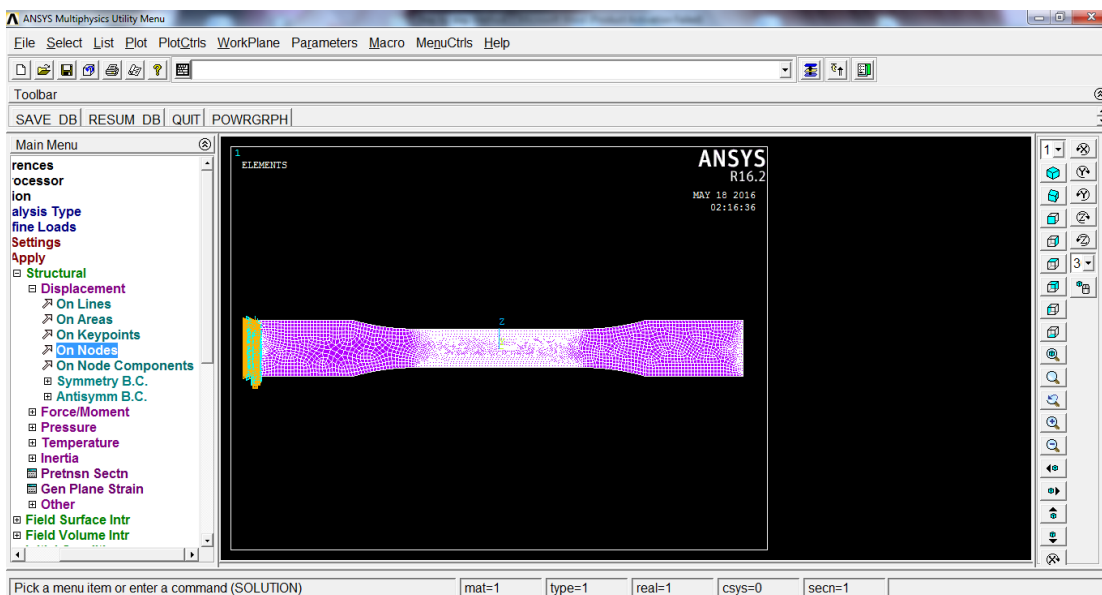


Figure 3.33: Result of Displacement Loading

Then, the “force/moment” from the structural is clicked and a small window will pop out by clicking the “on node”. At here, the box is ticked and choose the part that is needed to be applied. The “ok” is clicked to confirm it.

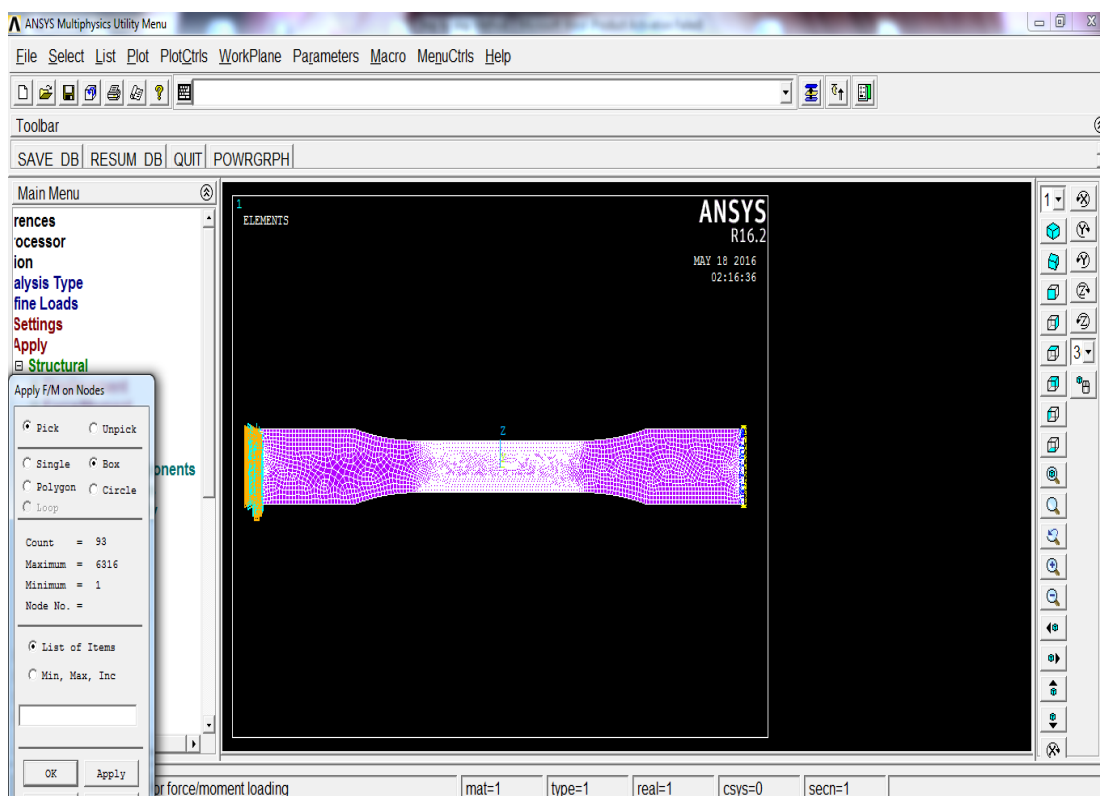


Figure 3.34: Defined of Force

After that, a window will pop out. At here, the direction of force/moment will be set to x-direction and the value of force is also set. Then, click “ok” to confirm it.

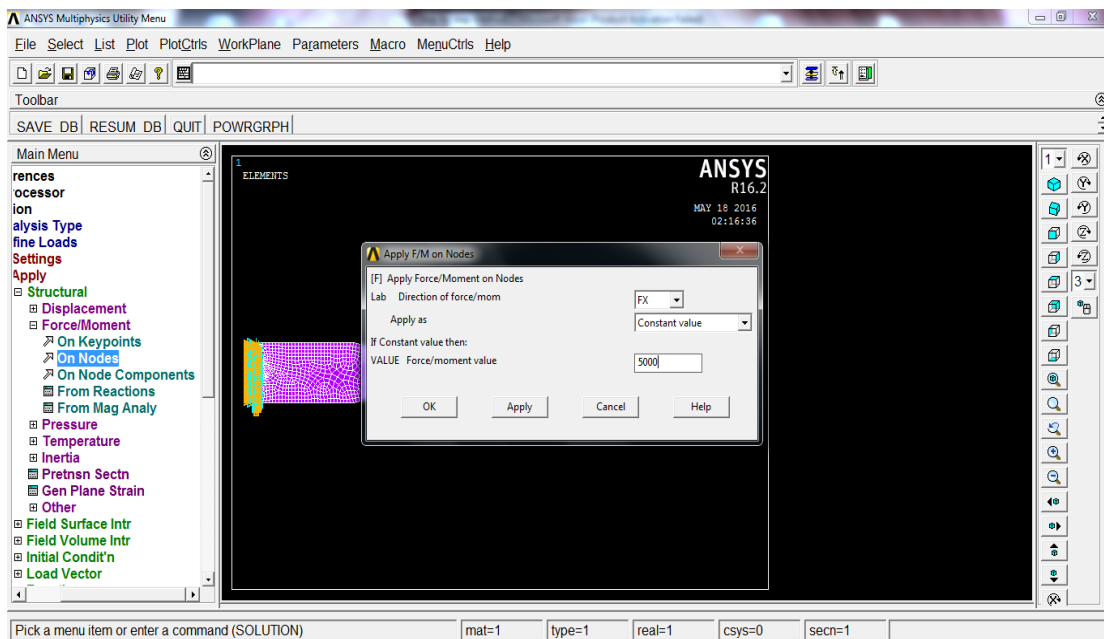


Figure 3.35: Setting of Force

After that, the result will be looked like in the figure 3.36 below.

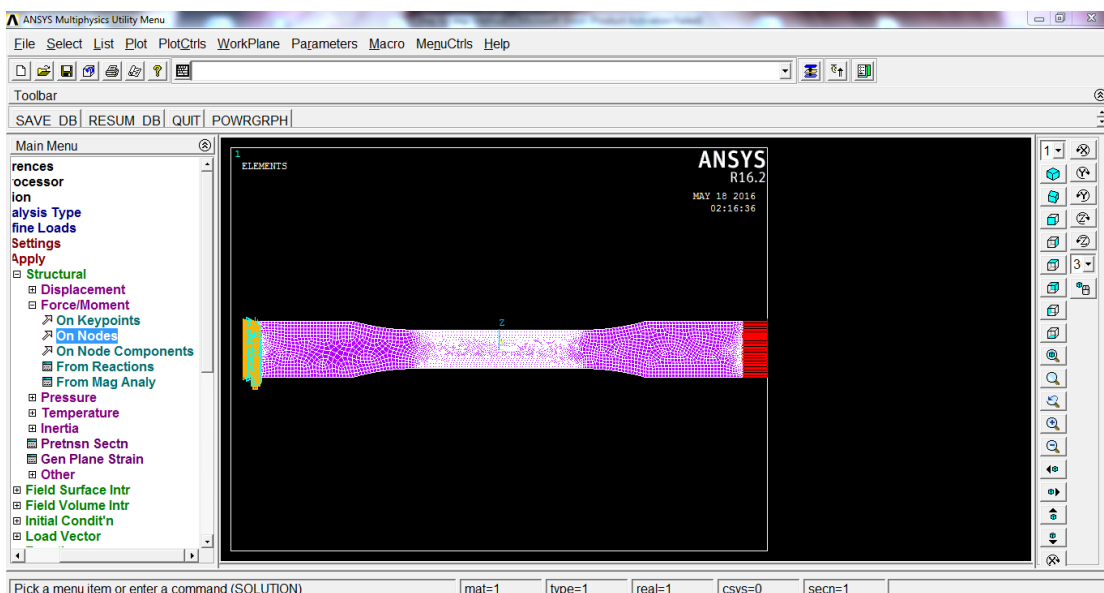


Figure 3.36: Result of Force Applied

Then, the “solve” from the solution is clicked and the “current LS” is chosen. After that, two small windows will pop out. The “ok” is clicked to start the solving process. Then, a window pop out to indicate that the solution is done.

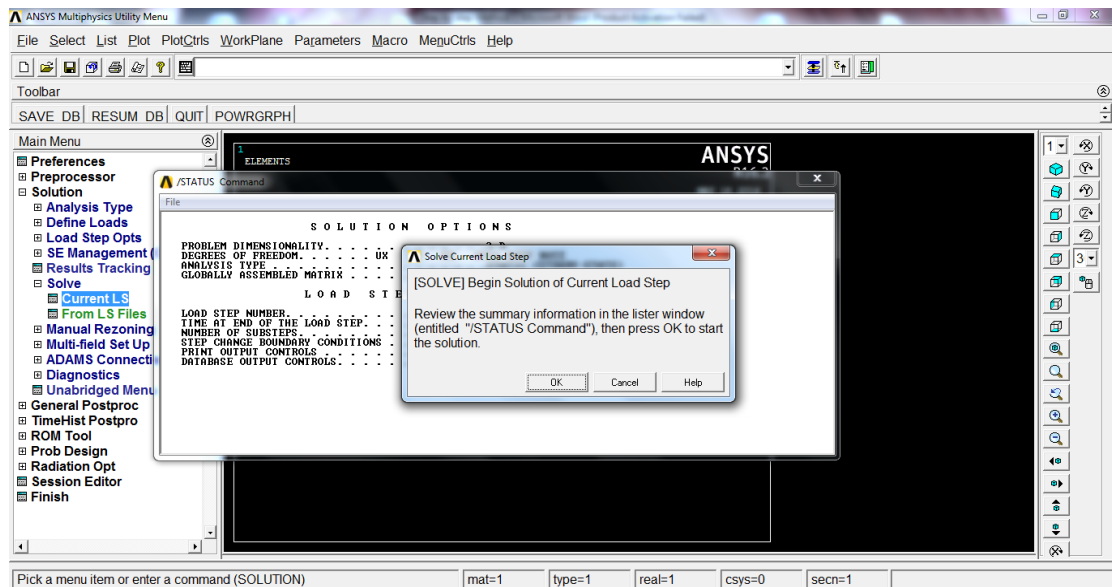


Figure 3.37: Window of Begin Solution

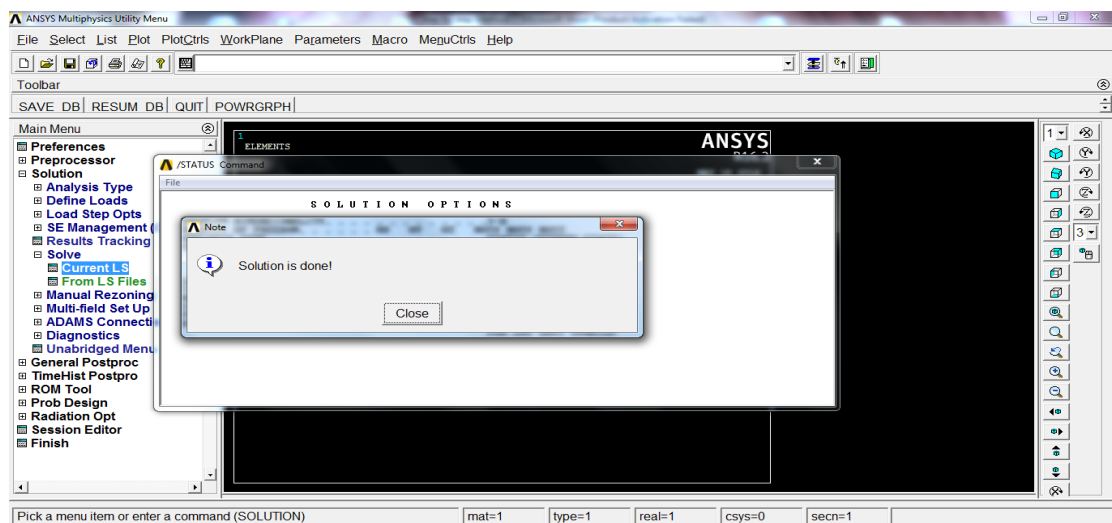


Figure 3.38: Window of Solution is Done

After that, the plot results is clicked. From there, the nodal solution is chosen. After clicked the “nodal solution”, a window will pop out. From there, we can choose what result from the nodal solution we want.

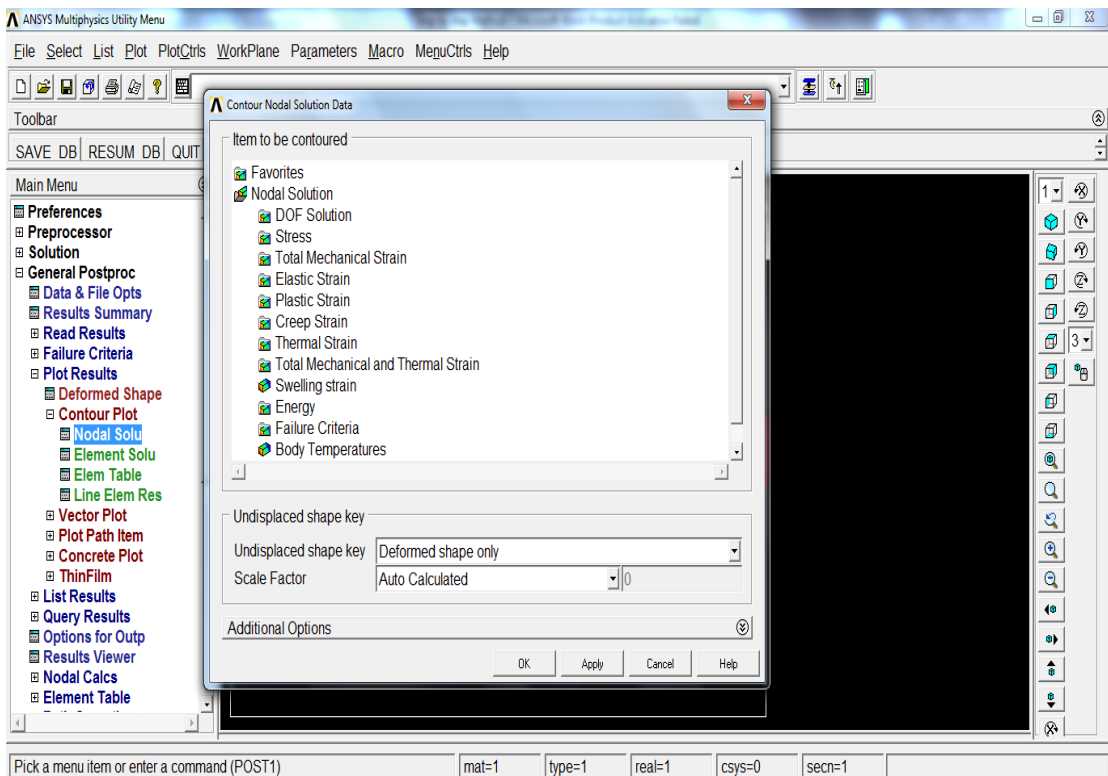


Figure 3.39: Window of Nodal Solution

At here, the x-component stress is chosen. Then click “ok” to confirm it. The result is shown at the figure 3.40 below.

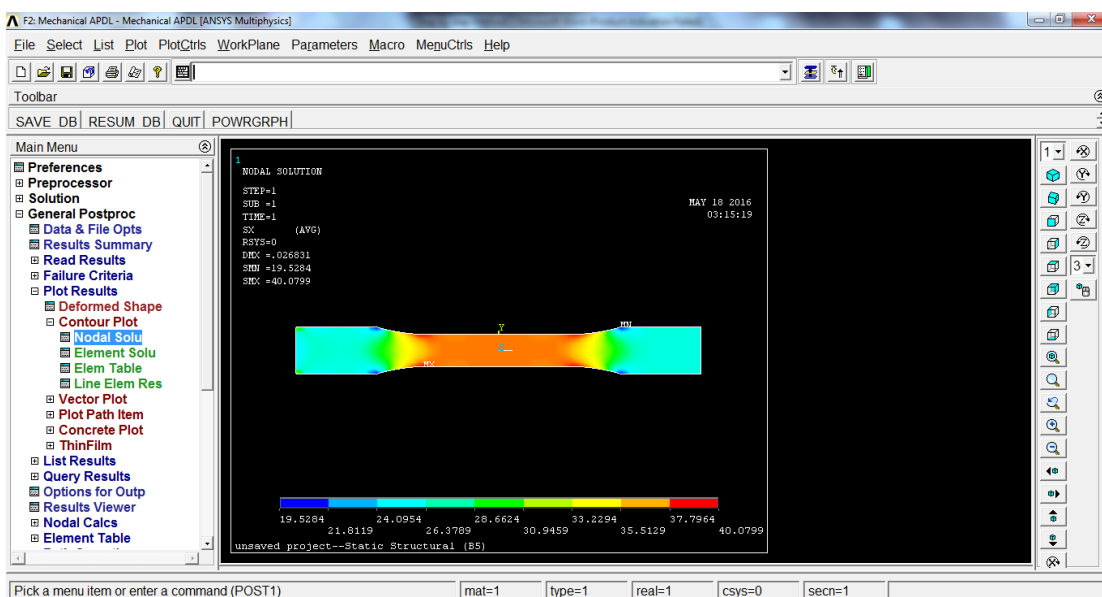


Figure 3.40: Analysis Results

After that, the timehist prospoc is clicked and a window of time history variable will pop out.

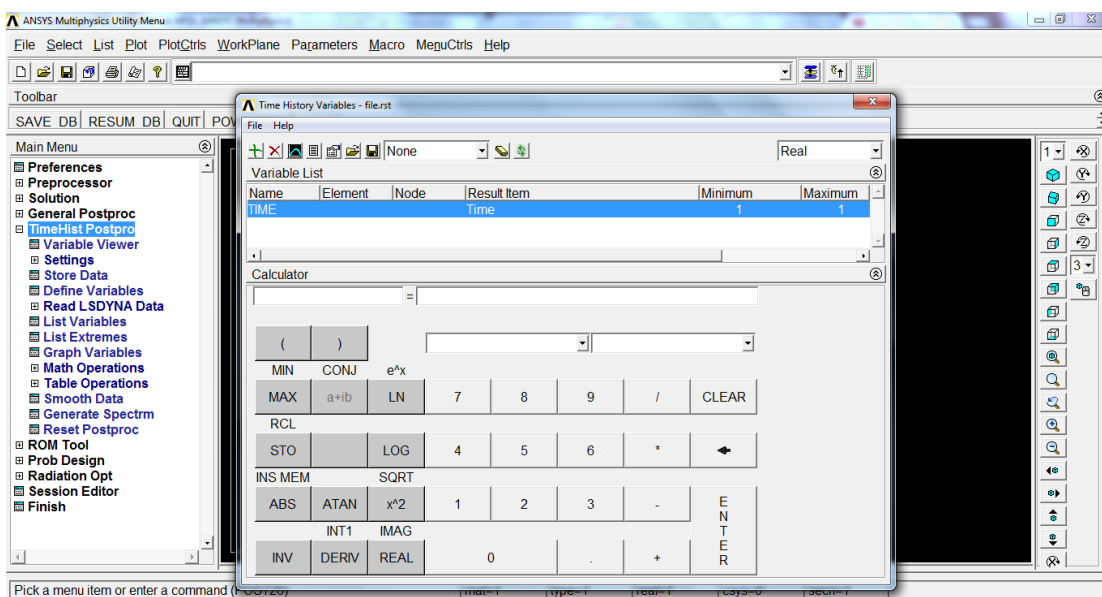


Figure 3.41: Window of Time History Variable

At here, the add data is clicked, and the nodal solution is chosen. Then the x-component stress is chosen. A window will pop out and a node is needed to be chosen. Then click the ok to confirm it. Then, the strain is chosen. A window will pop out and the same node is needed to be chosen. Then click the ok to confirm it.

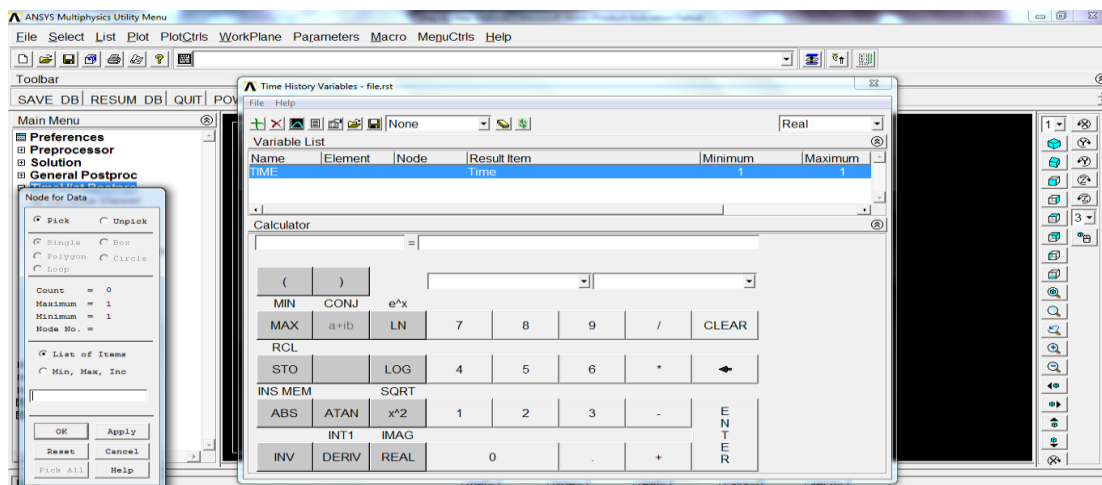


Figure 3.42: Window of Adding Nodes Data

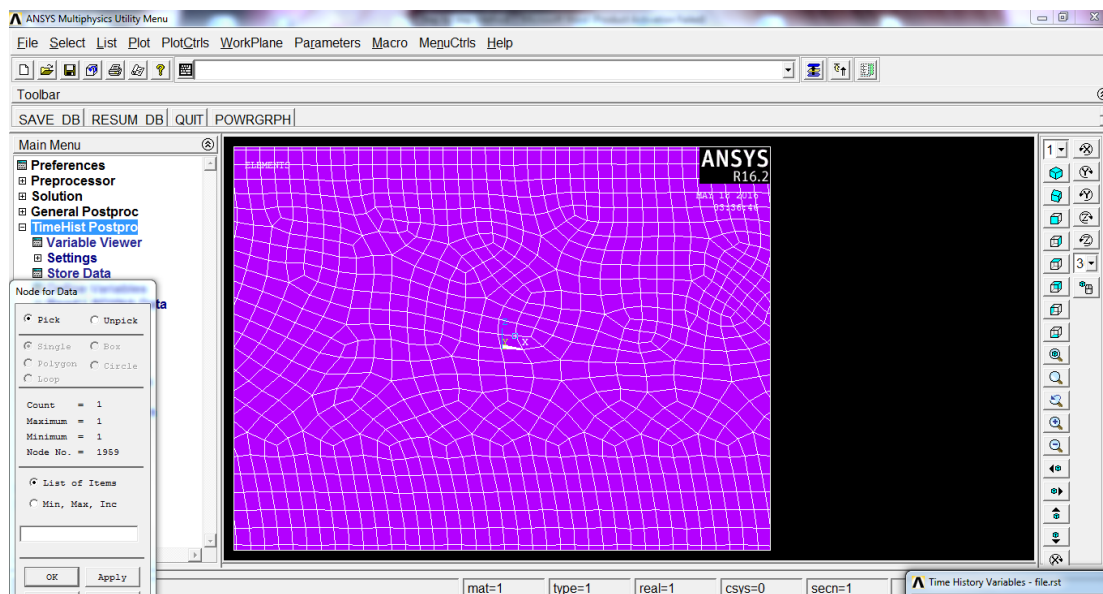


Figure 3.43: Window of Choosing Nodes

After that, the strain data is clicked as the x-axis. Then the graph data is clicked, the graph will be shown at the figure 3.44 below.

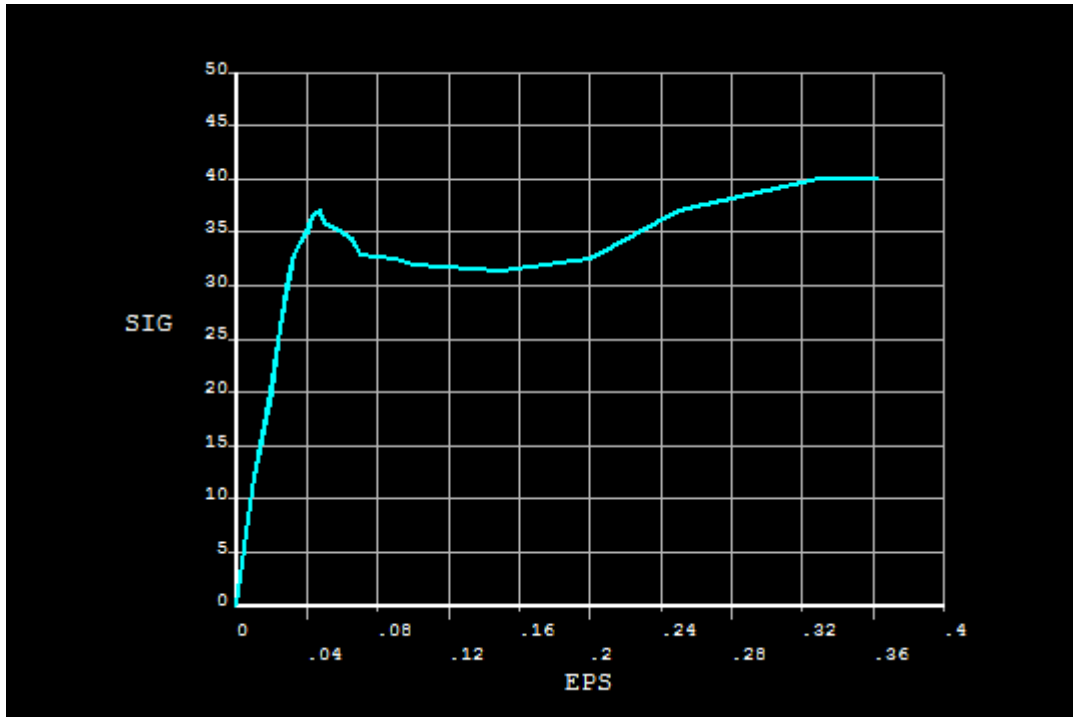


Figure 3.44: Results of Stress Strain Curve

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

This chapter provides all the results of the uniaxial tensile test analysis from simulation. The results from the simulation and the stress strain curve will be shown in this chapter. Besides that, the table with the list of stress and strain will also be shown in this chapter. All the results will be interpreted to understand the service behavior of the material.

4.2 UNIAXIAL STRESS STRAIN ANALYSIS

The procedures to perform the uniaxial stress strain analysis to get the service behavior were described in the methodology and the results were discussed. The dimension of testing specimen was followed standard ASTM D638. The element chosen is shell element 63. The dimension of standard ASTM D638 is shown in the figure 4.1 below.

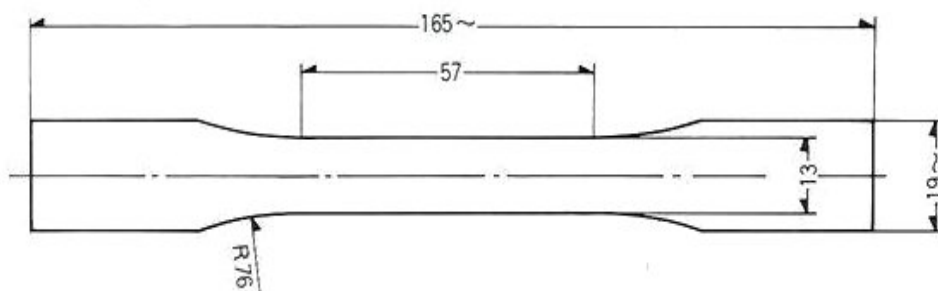


Figure 4.1: Dimension of testing specimen ASTM D638

The shell element is commonly used when analysis thermoplastic material, this due to large deflection capabilities are included in it. Besides that, based on the study, the shell element 63 is recommended when the thickness of the model is very thin. The thickness of this element type is assumed to be varied very smoothly over the area of the element with the thickness input at the four nodes. If the thickness is constant, only one TK(I) need be input. On the other hand, if the thickness is not constant, all four thicknesses must be input. The image of shell element 63 is shown in the figure 4.2 below. The input summary of the element shell 63 is shown in table 4.3 and table 4.4.

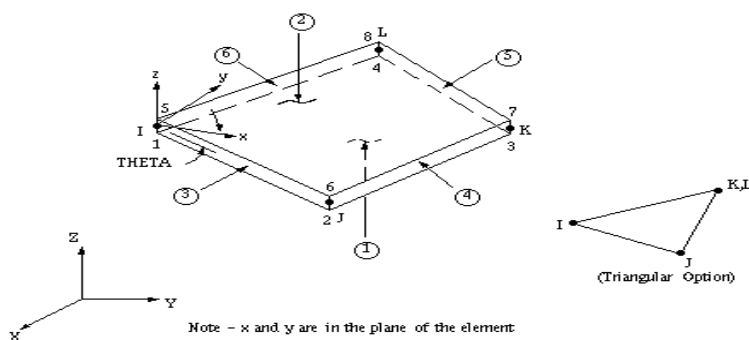


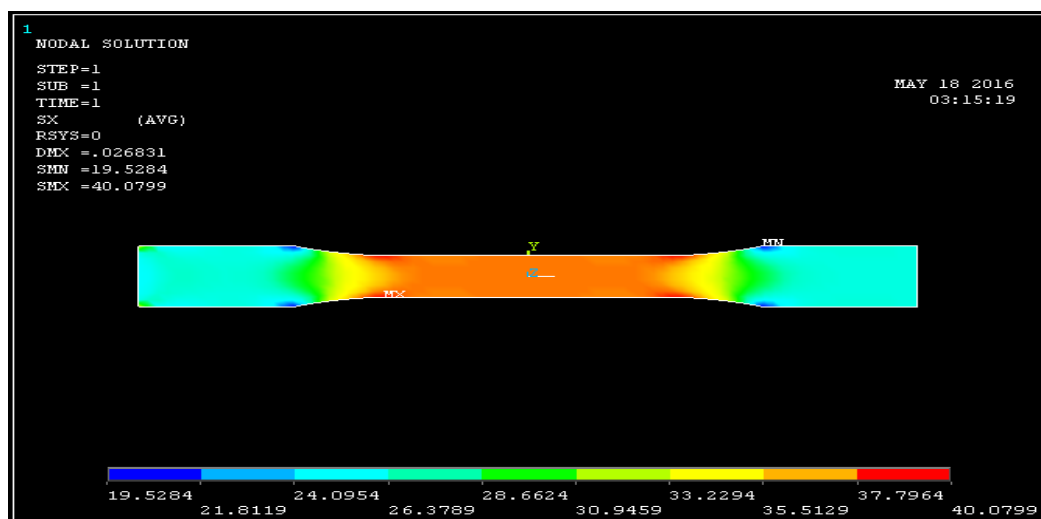
Figure 4.2: Image of element shell 63

Table 4.1: Input summary of element shell 63

Element Name	SHELL63
Nodes	I, J, K, L
Degrees of Freedom	UX, UY, UZ, ROTX, ROTY, ROTZ
Real Constants	TK (I), TK (J), TK (K), TK (L), EFS, THETA, RMI, CTOP, CBOT, (Blank), (Blank), (Blank), (Blank), (Blank), (Blank), (Blank), (Blank), (Blank), ADMSUA
Material Properties	EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ, DENS, GXY, DAMP
Surface Loads	Pressures: face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction), face 3 (J-I), face 4 (K-I), face 5 (L-K), face 6 (I-L)
Body Loads	Temperatures: T1, T2, T3, T4, T5, T6, T7, T8
Special Features	Stress stiffening, Large deflection, Birth and death
KEYOPT(1)	0 - Bending and membrane stiffness 1 - Membrane stiffness only 2 - Bending stiffness only

Table 4.2: Input summary of element shell 63 (cont)

KEYOPT(2)	0 - Use only the main tangent stiffness matrix when NLGEOM is ON. (Stress stiffening effects used in linear buckling or other linear prestressed analyses must be activated separately with PSTRES,ON .) 1 - Use the consistent tangent stiffness matrix (i.e., a matrix composed of the main tangent stiffness matrix plus the consistent stress stiffness matrix) when NLGEOM is ON and when KEYOPT(1)=0. (SSTIF,ON will be ignored for this element when KEYOPT(2)=1 is activated.) Note that if SOLCONTROL is ON and NLGEOM is ON, KEYOPT(2) is automatically set to 1; i.e., the consistent tangent will be used.
KEYOPT(3)	0 - Include extra displacement shapes, and use spring-type in-plane rotational stiffness about the element z-axis (the program automatically adds a small stiffness to prevent numerical instability for non-warped elements if KEYOPT(1)=0). 1 - Suppress extra displacement shapes, and use spring-type in-plane rotational stiffness about the element z-axis (the program automatically adds a small stiffness to prevent numerical instability for non-warped elements if KEYOPT(1)=0). 2 - Include extra displacement shapes, and use the Allman in-plane rotational stiffness about the element z-axis. See Section 14.43.6 of the <i>ANSYS Theory Reference</i> .
KEYOPT(5)	0 - Basic element printout 2 - Nodal stress printout
KEYOPT(6)	0 - Reduced pressure loading (must be used if KEYOPT(1)=1) 2 - Consistent pressure loading
KEYOPT(7)	0 - Consistent mass matrix 1 - Reduced mass matrix
KEYOPT(8)	0 - "Nearly" consistent stress stiffness matrix (default) 1 - Reduced stress stiffness matrix
KEYOPT(9)	0 - No user subroutine to define element coordinate system 4 - Element x-axis located by user subroutine USERAN (see the <i>Guide to ANSYS User Programmable Features</i> for user written subroutines)

**Figure 4.3:** The uniaxial stress strain tensile simulation result

The uniaxial stress strain tensile simulation result is shown in the figure 4.3 above. Since the load is applied in the x component, the x component of stress is chosen as the result. In the figure, we can see the maximum stress is 40.0799Mpa while the least is 19.5204Mpa. The stress is concentrated at the middle part, which means the deformation is occurred at the middle part.

4.3 THE STRESS STRAIN CURVE

The uniaxial stress strain tensile simulation result in the figure 4.4 can be plotted into the graph. The results from the simulation will be plotted into stress versus strain curve. From the stress strain curve, it explained the relationship between the stress and strain for a particular material. It is different for every material, it is determined by record the amount of deformation at distinct intervals of tensile stress. The stress strain curve able explains many properties of materials. The figure 4.4 below will show the result of the stress strain curve. The figure 4.5 will show the interpretation the stress strain curve.

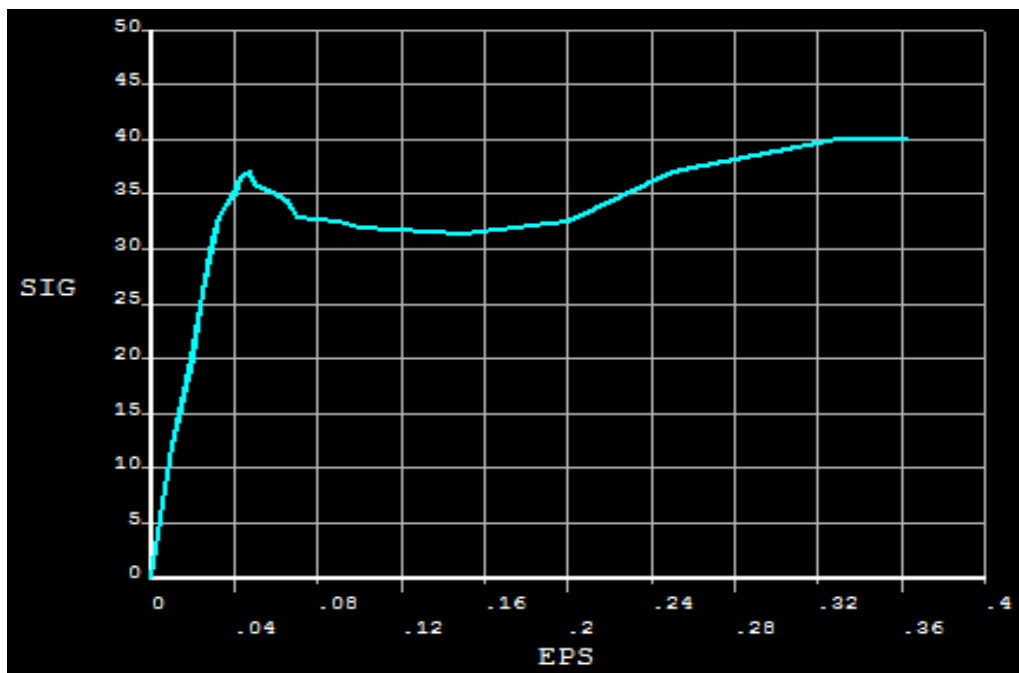


Figure 4.4: The stress strain curve

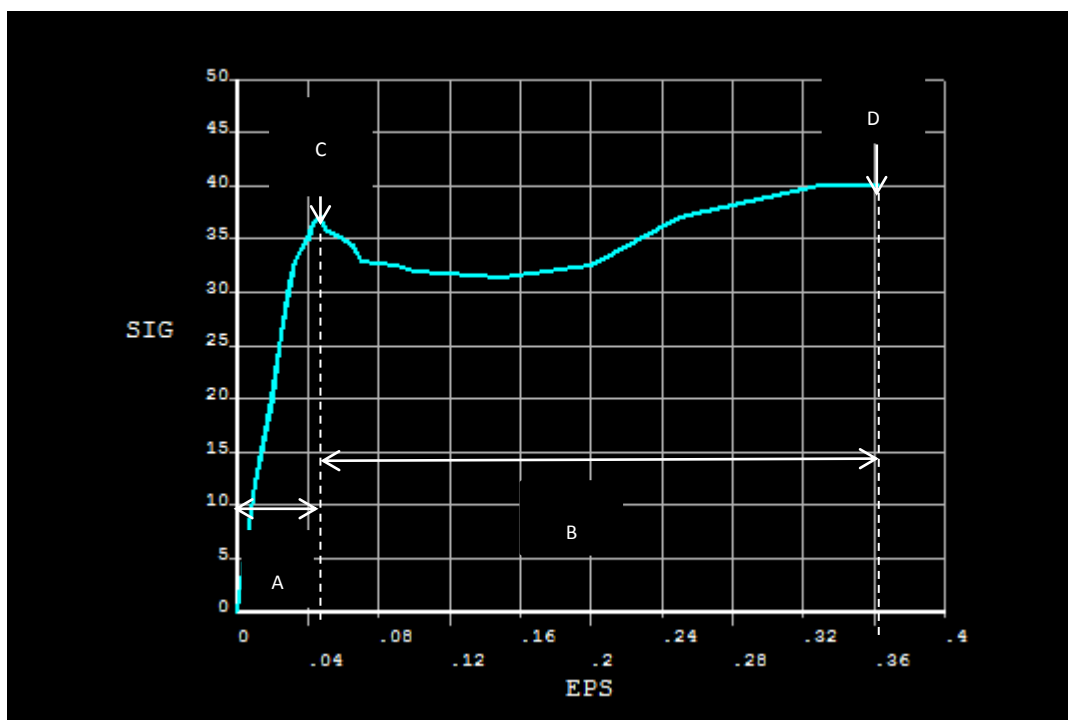


Figure 4.5: The interpretation of stress strain curve

Point A from the figure 4.5 above shows the elastic region. The elastic region of the stress-strain curve is the region in which the material will return to the undeformed state when applied forces are removed. The slope of the line in this region where stress is proportional to strain and is called the modulus of elasticity or Young's modulus. The modulus of elasticity defines the properties of a material as it undergoes stress, deforms, and then returns to its original shape after the stress is removed. It is a measure of the stiffness of a given material. To compute the modulus of elasticity, simply divide the stress by the strain in the material. Since strain is unitless, the modulus will have the same units as the stress.

Point B from the figure 4.5 above shows the plastic region. The plastic region is the portion where some permanent deformation will occur, even if the load is removed. It produces a permanent change in the shape and size of a solid body without fracture, resulting from the stress beyond the elastic limit. The difference between elastic region and plastic region is material will return to the original state when applied forces are removed in elastic region while the material will undergo some permanent deformation in the plastic region although the applied forces are removed.

Point C from the figure 4.5 above shows the yield point. In fact, to reach plastic deformation material always have to begin with elastic deformation. At this yield point, the stress-strain curve deviates from the straight-line relationship and Hooke's Law no longer applies as the strain increases faster than the stress. From this point in the tensile test, some permanent deformation occurs in the specimen and the material is said to react plastically to any further increase in load. The material will not return to its original, unstressed condition when the load is removed.

Point D from the figure 4.5 above shows the ultimate tensile strength. It is the highest point inside the stress strain curve. Ultimate tensile strength is measured by the maximum stress that a material can withstand while being “stretched or pulled before breaking.

Table 4.3: List of stress and strain

Curve Points	Stress (MPa)	Strain
1	0.00	0.000
2	12.00	0.010
3	20.00	0.020
4	26.00	0.025
5	31.00	0.030
6	33.00	0.033
7	35.00	0.040
8	36.50	0.043
9	37.00	0.047
10	36.00	0.050
11	35.00	0.060
12	34.50	0.065
13	33.00	0.070
14	32.50	0.090
15	32.00	0.100
16	31.50	0.150
17	32.50	0.200
18	37.00	0.250
19	39.00	0.300
20	40.00	0.330

Table 4.3 above shows the list of stress and strain obtained from the stress strain curve. The yield point is at curve point 7, when the stress is at 37.00MPa while the ultimate tensile strength is at curve point 20, when the stress is at 40.00Mpa.

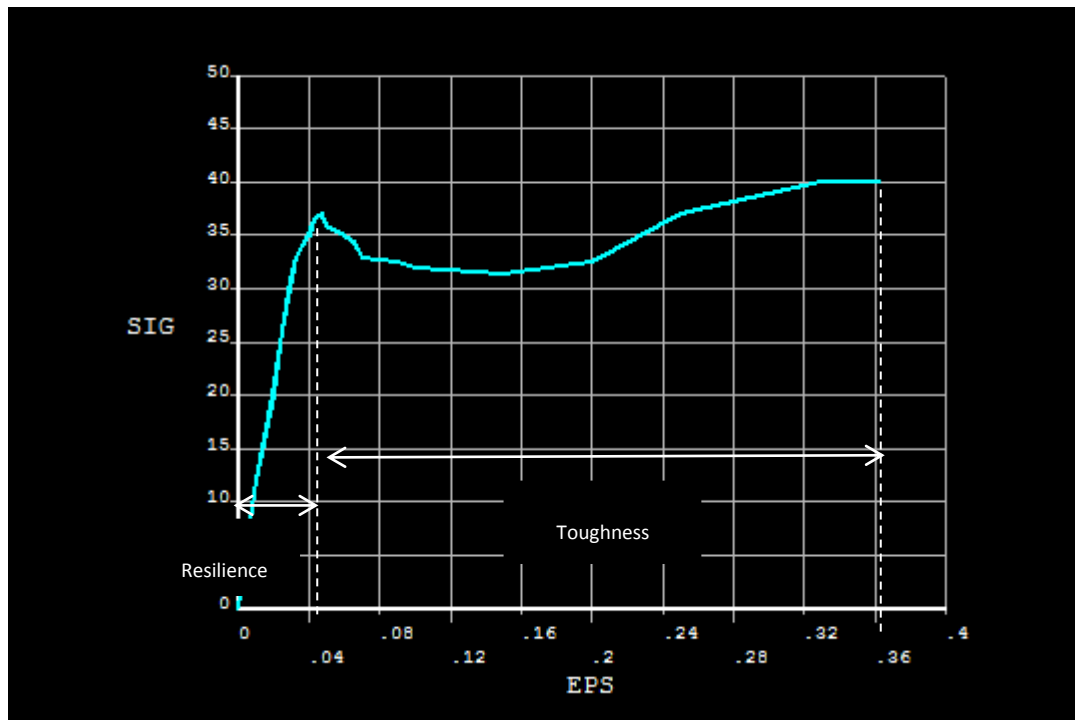


Figure 4.6: Resilience and toughness of material

The figure 4.6 above shows the resilience and toughness of the material. The area up to the yield point is the modulus of resilience, and the total area up to fracture is termed the modulus of toughness as shown in figure 4.6. The term resilience alludes to the concept that up to the point of yielding, the material is unaffected by the applied stress and upon unloading will return to its original shape. But when the strain exceeds the yield point, the material is deformed irreversibly, so that some residual strain will persist even after unloading. The modulus of resilience is the quantity of energy the material can absorb without suffering damage. In contrast, the modulus of toughness is the energy needed to completely fracture the material. Materials showing good impact resistance are generally those with high modulus of toughness.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 INTRODUCTION

This chapter presents a summary of the findings in this research report and conclusions for the same. In addition the chapter offers recommendations for further research in a similar field.

5.2 CONCLUSIONS

In conclusion, the service behavior of polypropylene has been determined through the uniaxial tensile test by using ANSYS APDL. It has a non-linear viscoelasticity behavior. This behavior lead to non-linearity in stress strain curve.

Viscoelasticity is the property of materials that exhibit both viscous and elastic characteristics when undergoing deformation. Viscoelasticity material is a strain rate dependence material. Viscoelastic materials respond differently depending on how fast they are stretched. The displacement is related to strain, so the strain rate defines how fast the material is stretched. Therefore, viscoelastic materials are said to be strain rate dependent. The same material has two very

different responses to a force depending on how fast the force was applied. Faster strain rates cause more elastic responses and the material behaves more like a solid (stiffer), whereas slower strain rates cause more fluid-like behavior. It is the common characteristic of viscoelastic material.

5.3 RECOMMENDATIONS

In future recommendation works, the experimental approach to find out the real work service behavior is suggested. In current project, the analysis work is only done through the simulation approach, it has the limitation that we cannot know how the material will react when it comes to the real world situation. So, by doing the experimental work, we can compare the simulation results with the experimental results to find out how much the difference between them to get the most accurate prediction of the properties of the material.

REFERENCE

This thesis is prepared based on the following references:

- [1] ASI magazines 2015. Market Trends: Automotive Industry Keeps Thermoplastic Elastomers Market Strong (online). <http://www.adhesivesmag.com/articles/93691-market-trends-automotive-industry-keeps-thermoplastic-elastomers-market-strong> (2 March 2015)
- [2] ING. Katarina Szeteiova. “Automotive Materials Plastics in Automotive Markets Today”, Slovak University of Technology Bratislava.
- [3] McGrum, N. G., Buckley, C. P. and Bucknall, C. B. 1997. “Principles of Polymer Engineering”, Oxford University Press, Second edition, New York.
- [4] Reh, S., Beley, J.-D., Mukherjee, S., & Khor, E. H. (2006). Probabilistic finite element analysis using ANSYS. *Structural Safety*, 28, 17–43.
- [5] Archer, G. C. (1996). Object-Oriented Finite Element Analysis, 1–285.
- [6] Arriaga, a., Lazkano, J. M., Pagaldai, R., Zaldua, a. M., Hernandez, R., Atxurra, R., & Chrysostomou, a. (2007). Finite-element analysis of quasi-static characterisation tests in thermoplastic materials: Experimental and numerical analysis results correlation with ANSYS. *Polymer Testing*, 26(3), 284–305.
- [7] No, R., Compagnia, T. D., Italiana, P., Rapporto, S., & Roma, M. I. (1996).
- [8] Rasplavov, S., Tarcy, G. P., Processing, E., Press, P., & Kalafut, P. E. J. (1990). United States Patent [191].
- [9] TESTRESOURCES. Making Dogbone Tensile Test Samples. Retrieved from <http://www.testresources.net/blog/making-dogbone-tensile-test-samples> (09 August 2013)

- [10] Barnard, B. M. (2012). Physical Properties, 1–3.
- [11] Yilmaz, H. (2010). Tensile strength testing of thin spray-on liner products (TSLs) and shotcrete. *Journal of the Southern African Institute of Mining and Metallurgy*, 110(October), 559–569.
- [12] ASTM. (2002). D 882: Standard Test Method for Tensile Properties of Thin Plastic Sheeting. *Astm*, 14, 1–10.
- [13] Udomphol, T. (2007). Tension Test, 1–12.
- [14] NDT RESOURCE CENTER. Tensile Properties. Retrieved from <https://www.ndeed.org/EducationResources/CommunityCollege/Materials/Mechanical/Tensile.htm>
- [15] Dr. Dmnitri Kopeliovich. SubstTech Substances & Technologies. Fracture Toughness. http://www.substech.com/dokuwiki/doku.php?id=fracture_toughness (10 July 2015)
- [16] Degarmo, E. Paul; Black, J T.; Kohser, Ronald A. (2003), *Materials and Processes in Manufacturing (9th ed.)*, Wiley, ISBN 0-471-65653-4.
- [17] NDT RESOURCE CENTER. Polymers. Retrieved from <https://www.ndeed.org/EducationResources/CommunityCollege/Materials/Introduction/polymers.htm>.
- [18] Materials, B. P., & Sensitivity, S. R. (2013). MAAE 3901 – Mechanical and Aerospace Engineering Carleton University, Ottawa ON Formal Laboratory Report I on “Experiment E : Properties of Polymer Materials” Tensile Properties of Ductile and Brittle Polymer Materials and Strain Rate Sensitivity.
- [19] Conjecture Corporation, 2013. (Online). <http://www.wisegeek.org/what-are-polymers.htm>.
- [20] Priscila Anadão (2012) INTECH opens science and open minds. *Polymer/ Clay Nanocomposites: Concepts, Researches, Applications and Trends for The Future*.

<http://www.intechopen.com/books/nanocomposites-new-trends-and-developments/polymer-clay-nanocomposites-concepts-researches-applications-and-trends-for-the-future> (27 September 2012)

[21] S.R.S. Serope Kalpakjian, “Polymers: Structure, General Properties, and Applications,” in Manufacturing Engineering and Technology, 6th ed., Prentice Hall, 2009.

[22] W. Callister, “Polymer Structure,” in Materials Science and Engineering, John Wiley and Sons., 2010, pp. 532-568.

[23] K. K. L. a. T. R. S. Ralph M. Paroli, “Thermoplastic Polyolefin Roofing Membranes,” 1999.

[24] Canonsburg, T. D. (2010). ANSYS Mechanical APDL Tutorials. Knowledge Creation Diffusion Utilization, 15317(November), 724–746.

