



Characterizing the Nonlinear Deformation of a Polypropylene Polymer.

Author: Kapil Bhandari

Instructor: Silas Zewdie Gebrehiwot

Degree Thesis
Materials Processing Technology
2021

DEGREE THESIS	
Arcada University of Applied Sciences	
Degree Programme:	Materials Processing Technology
Identification number:	22673
Author:	Kapil Bhandari
Title:	Characterizing the Nonlinear Deformation of a Polypropylene Polymer.
Supervisor (Arcada):	Silas Zewdie Gebrehiwot
Commissioned by:	
<p>Abstract:</p> <p>A polymer has both elastic and plastic region in stress strain curve. Material modelling is the process of modelling the stress strain behaviour using finite element method by computational software's. Material models help to define relationship between stress and corresponding strain at any particular time period. There are several material models to describe the relation, some of them are Ramberg's Osgood model, Uniaxial model, Power law, Ludwik's model. when the applied load surpasses the yield strength then material starts to deform. Factors like time, temperature, material properties, load etc, affects the deformation. But only the time deformation using prescribed displacement under room temperature is analysed using Ramberg's Osgood and Uniaxial material model of COMSOL. Using a Testometric machine, tensile test of a polypropylene as a thermoplastic was conducted. A sample piece(dog-bone) according to ASTM D368 type 1 of a polypropylene was used as a specimen. A prescribed displacement of 20mm/min was set for each test and a test lasted for approximately 21 minutes each. From the obtained data the stress strain graph was plotted. From stress strain graph the values of young's modulus, Poisson's ratio, strain hardening exponent, reference stress and strain were calculated. A time dependent model was modelled in COMSOL. From the Nonlinear module Ramberg's Osgood and Uniaxial model was selected. The calculated values from experiment were substituted in each model. The obtained result was analysed using graphical observation. Deviation and Error analysis was done to find which model could best describe the behaviour.</p> <p>During this thesis it was found; the result obtained from the Uniaxial material model to be close to experiment in compared to Ramberg's Osgood. It should be noted that multiple temperature and strain rates are used to fully define the material behaviours. For other condition the next material model might give the best result. As expected, the experimental work is very much similar to the past research conducted on characterizing the nonlinear deformation of polypropylene.</p>	
Keywords:	Nonlinear elasticity, Stress, Strain, Polypropylene, Uniaxial, Young's modulus, Deviation, Material Model, Polymer, Finite Element Method, ASTM.
Number of pages:	48
Language:	English
Date of acceptance:	

CONTENTS

1	INTRODUCTION	1
1.1	Background	1
1.2	OBJECTIVES	2
2	LITERATURE REVIEW	3
2.1	Introduction	3
2.2	Stress, Strain and Young's Modulus	4
2.2.1	<i>Normal Stress(σ)</i>	4
2.2.2	<i>Normal Strain(ϵ)</i>	5
2.2.3	<i>Young's Modulus(E)</i>	8
2.3	Uniaxial Tensile Testing	9
2.3.1	<i>Importance of uniaxial test</i>	10
2.4	Linear Elasticity; Hooke's Law	11
2.4.1	<i>Hooke's Law</i>	11
2.4.2	<i>Poisson's Ratio</i>	12
2.5	Stress- strain relations	12
2.5.1	<i>Elastic Limit or Yield Point</i>	13
2.5.2	<i>True strain Vs Engineering Strain</i>	14
2.6	Elastic Deformation and Plastic Deformation	15
2.7	Plasticity Theory	16
2.8	Finite Element Method(FEM)	18
2.8.1	<i>Linear vs Nonlinear</i>	20
2.9	COMSOL	20
2.10	Non-Linear Elasticity Model	21
2.10.1	<i>Advantages</i>	22
2.10.2	<i>Disadvantages</i>	22
2.10.3	<i>When to use material non-linearity</i>	22
3	REVIEW OF METHODS	22
3.1	Analytical Methods	23
3.2	Finite Element Method using Numerical approach	24
3.3	Finite Element Methods using Software's	27
3.3.1	<i>Modelling Geometry through Finite Element method</i>	27
3.4	Calculating the parameters required for modelling	29
3.4.1	<i>Linear Modelling</i>	29
3.4.2	<i>Non- Linear Modelling with Uniaxial Material Model</i>	30

3.4.3	<i>Non- Linear Modelling with Ramberg-Osgood Material Model</i>	30
4	EXPERIMENT	30
4.1	Poisson's ratio calculation	32
5	COMSOL MODELLING	33
5.1	UNIAXIAL MATERIAL MODEL	38
5.2	RAMBERG OSGOOD MODEL	41
5.2.1	<i>Young's modulus calculation</i>	41
5.2.2	<i>Reference stress strain calculation</i>	42
5.2.3	<i>Stress exponent(n)</i>	42
6	RESULT	44
6.1	Ramberg's Osgood material model.....	44
6.2	Uniaxial material model	45
7	DISCUSSION	46
8	CONCLUSION	47
9	REFERENCES	49

Figures

Figure 1 Elongation of Bar	5
Figure 2 Axially loaded body elongation(Kassir, 2017)	6
Figure 3 Axially loaded member internal forces(Kassir, 2017)	7
Figure 4 Uniaxial Tensile Test Diagrammatic representation(Davis, 2004).....	9
Figure 5 Stress Strain Relationship of ductile material	13
Figure 6 Engineering Strain Vs True Strain	14
Figure 7 Plasticity modelling(Veerappan,2017).....	17
Figure 8 Linear and Nonlinear graph	20
Figure 9 Algorithm of process.....	23
Figure 10 One dimensional bar	25
Figure 11 Finite discretization of one-dimensional bar.....	25
Figure 12 ASTM D368 Type 1 parameters	31
Figure 13 Setup for tensile test(ARCADA B225).....	31
Figure 14 Tensile specimen before and after test.....	32
Figure 15 Tensile test (Stress vs Strain) graph.....	32
Figure 16 Calculation of Poisson's ratio	33
Figure 17 Modelling of Tensile Specimen in Solidworks2020.....	33
Figure 18 Parameters for defining the values in COMSOL	34
Figure 19 Describing the Displacement	34
Figure 20 Selection of Mesh.....	35
Figure 21 Final Meshed Geometry(COMSOL).....	35
Figure 22 Defining the time boundary	35
Figure 23 Changing from physics controlled to user controlled relative tolerance.....	36
Figure 24 Enabling the Direct Solver	36
Figure 25 Selecting Automatic(Newton) solver.....	37
Figure 26 Calculating the Stress Strain values	37
Figure 27 Parameters for uniaxial model	38
Figure 28 Interpolation curve with real stress strain data.....	39
Figure 29 Obtained stress strain graph using real data.....	39
Figure 30 Data fitting for uniaxial model.....	40
Figure 31 Interpolation curve using fitted data	40
Figure 32 Obtained stress strain graph using fitted data	41

Figure 33 Youngs modulus calculation graph.....	41
Figure 34 Reference stress strain calculation graph	42
Figure 35 Stress exponent calculation graph.....	42
Figure 36 Defining Ramberg's Osgood parameters	43
Figure 37 Stress strain graph obtained from Ramberg's Osgood model	43
Figure 38 Stress Displacement graph obtained from Ramberg's Osgood model	44
Figure 39 Comparison of Ramberg's Osgood model with tensile result	44
Figure 40 COMSOL stress distribution for Ramberg's Osgood model	45
Figure 41 Comparison of COMSOL uniaxial model with tensile result.....	45
Figure 42 COMSOL stress distribution for uniaxial model	46
Figure 43 Deviation and Error analysis.....	46

Abbreviation

FEM: Finite Element Method

FEA: Finite Element Analysis

MATLAB™: Matrix Laboratory

HF: Hardening Function

ASTM: American Society for Testing and Materials.

ISO: International organization for Standardization.

Symbols

σ = Normal Stress

A = Area

F = Resisting Force

P = Applied Force

ϵ = Extensional Strain

L^* = Original Length

L = Increased Length

ΔL = Change in Length

E = Young's Modulus

σ_1 = Initial Stress

σ_2 = Final Stress

ϵ_1 = Initial Strain

ϵ_2 = Final Strain

σ_1 = Initial Stress

σ_2 = Final Stress

$\epsilon_1 = \text{Initial Strain}$

$\epsilon_2 = \text{Final Strain}$

$\nu = \text{Poisson's Ratio}$

$F_v = \text{Body force per volume}$

$\rho = \text{Density}$

$\ddot{u} = \text{Acceleration}$

$C = \text{Fourth - order tensor}$

$:$ = Denotes the tensor product

λ and $\mu = \text{Lame's constants}$

$n = \text{Stress exponent}$

$\sigma_t = \text{True Stress}$

$\sigma_y = \text{Stress at Yield Point (Yield Stress)}$

$k = \text{Strength Coefficient}$

$\epsilon_t = \text{True strain}$

1 INTRODUCTION

1.1 Background

An elastic body or a material is called linear elastic or Hookean; if the force needed to compress or extend by a certain distance is directly proportional to that distance. To describe the linear elasticity of a homogeneous isotropic material, only two parameters are needed i.e., Young's Modulus and the Poisson ratio. These values can be derived from simple uniaxial tension or compression test. For engineering application all the designs are done in this point. However, many applications require understanding of broad strains, with deformation that are fundamentally nonlinear and stresses that are dependent on the underlying material properties (Mihai & Goriely, 2017:p.1).

Knowledge of strength is becoming increasingly important due to increased use, ranging from small part to large structure. Mechanics of deformation leads the engineer into better understanding the conditions leading to failure. From a structural point of view, a better understanding of this permits a better design (Schröder *et al.*, 2017).

In all disciplines of science and engineering, the focus is on what will happen when a system of interest is subjected to the effect of the environment. Anything that happens can be described mathematically, in the sense that we can formulate a set of mathematical expressions that describes the process, allowing engineers to determine how the process occurs in space and time. Using simulation software's, these phenomena can be observed without performing that operation. (Koutromanos, 2018:p.1) . Testing in laboratory is both expensive and time consuming (Hawkins, 2007:pp.179–194).

While it is possible to study the elastic indentation of a material by using the analytical method, elastic-plastic indentation being sufficiently complex requires a finite element technique. The small increment in a certain variable can be accurately calculated from the instantaneous time of change using FEM. The Problems are solved by discretization in space dimension, implemented by constructing mesh. The solution is achieved in the numerical domain which has finite numbers of point. (Anandarajah, 2011) .

The tensile test is one of the commonly used tests for evaluating material. The specimen is gripped at the two ends and the load is applied. The applied load gradually leads to elongation and eventually fracture of specimen. The applied force and elongation are

recorded and plotted to get the material properties. The obtained material characteristics from the test are used for quality control in production, evaluation of newly developed materials, and static-strength requirement of designs(Davis, 2004) .

Polypropylene is one the most extensively used thermoplastics by the automotive industry. Typical examples from this field are dashboards and bumpers. It is also one of the most versatile polymers with virtually applied in all end-used products. The limiting feature of polypropylene is their inelastic behaviour which changes with loading rate effect. This loading rate effect changes the strength and stiffness of material (Szpieg, Giannadakis & Varna, 2011:pp.625–652).

On the other hand, the main tool commonly used for designing such parts is the finite element method. A material model capable of representing the most significant properties of the thermoplastic at hand is required for accurate numerical forecasts of the response generated. The prediction of nonlinear behaviour with available finite element codes is a challenging task. Obviously, development of suitable model for polypropylene demands good knowledge of the mechanical behaviour with mechanisms at the meso-scale (Mihai & Goriely, 2017:p.1).

1.2 OBJECTIVES

The major objectives of this thesis are :

- (1) To characterize the stress-strain behaviour of polypropylene using Uniaxial material model of COMSOL and compare it with the results obtained in the laboratory.
- (2) To characterize the stress-strain behaviour of polypropylene using Ramberg-Osgood material model of COMSOL and compare it with the results obtained in the laboratory.

2 LITERATURE REVIEW

2.1 Introduction

Polypropylene is one of the commonly used thermoplastics by human mankind. The extensive use is found in automotive industry. The reason can be attributed for cheap production, can be reprocessed several times and low environmental impact (Anon, n.d.).

All the physical observable things around us are material and their behaviours are non-linear in real life (Mihai & Goriely, 2017:p.20170607).

The applied external load to a structural component result in internal forces and deformation. The objective of this thesis is to determine the induced stress and strains, instead of focusing on deformation and internal forces. This is because the effect of load in specimen is directly proportional to the cross-sectional area. In addition, Changes in angle and length are measured relative to their instantaneous values. Using continuum mechanics theories, elastic and plastic behaviour are considered. The general mechanism is that when the applied load is removed if the body returns to its initial original shape, then it is called as an elastic deformation. The behaviour is called plastic if permanent deformation occurs(Kassir, 2017) . The behaviour of either plastic or elastic is studied by the three different aspects:

(1) Equilibrium:

When the applied external force and internal forces are in equilibrium then the body is said to be in state of equilibrium which requires the stresses to satisfy six equations of statics in space. This includes three equation of moment equilibrium and three equation of force equilibrium(Kassir, 2017) .

(2) Geometry of deformation:

When the load is applied, structure deforms which leads in change in shape and volume of object. The nature of deformation is considered in determining the response of load to external force. In continuum mechanics relation between displacement and strain forms the governing equation for deformation(Kassir, 2017) .

(3) Mechanical behaviour of the material:

The third approach is laboratory which involves the relation between strain and stress by standard tests. For elastic bodies Hooks law governs up to and including

yield point. Beyond the yield stress hardening occurs until ultimate stress and material failure. Various mathematical models are found to predict the plastic behaviour(Kassir, 2017:pp.1–2) .

2.2 Stress, Strain and Young’s Modulus

2.2.1 Normal Stress(σ)

When mechanical force is applied on a body, the body gets deformed. Due to deformation, there is change in intermolecular force inside the molecule. Developing some opposing forces within the body’s volume with respect to the applied deforming forces. Stress is the resisting force per unit area. Mathematically it is written as (Schröder *et al.*, 2017):

$$\sigma = \lim_{\Delta A \rightarrow 0} \frac{\Delta F}{\Delta A} \quad (1)$$

The above equation suggests that; for different area, the value of stress varies with the point. The equation can be re written as:

$$\sigma = \frac{\delta F}{\delta A} \quad (2)$$

Rearranging the equation(2) we get,

$$\delta F = \delta A * \sigma \quad (2.1)$$

On integrating on both sides, we get,

$$\int \delta F = \int \delta A * \sigma \quad (2.2)$$

Since sigma is not constant with respect to cross section. Using concept of average stress

$$\sigma_{Avg} = \text{Average stress}$$

Now on integrating and rearranging the equation (2.2) we get,

$$\sigma = \frac{F}{A} \quad (2.3)$$

From condition of static equilibrium, resisting force(F) equals to the applied force(P). Then equation (2.3) can be written as:

$$\sigma = \frac{P}{A} \quad (2.4)$$

The standard unit of stress in international system of units (SI) System is $\frac{N}{m^2}$. Stress is neither vector nor scalar quantity. It is a tensor quantity. In general tensor is a vector quantity which does not follow vector addition but follows simple algebraic addition. To define stress three components are needed which are Magnitude, Direction, and plane on which they are acting. The positive value for σ is due to pull force on which it acts whereas negative value of stress indicates compressive stress (Schröder *et al.*, 2017).

2.2.2 Normal Strain(ϵ)

Strain is measure of deformation. Strain can be measured and observed physically. Mathematically strain is change in dimension over original dimension. Since, strain is the ratio of dimension so, it is dimensioning less term. For uniaxial strain state (Hibbeler, 2018:pp.92–93)

$$\epsilon_{avg} = \frac{\Delta L}{L} = \frac{L^* - L}{L} \quad (2.5)$$

$L^* = \text{Original Length}$

$L = \text{Increased Length}$

$\Delta L = \text{Change in Length}$

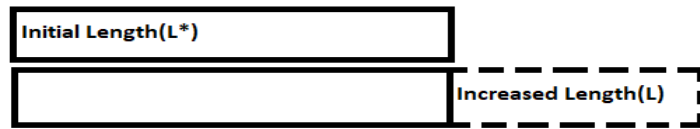


Figure 1 Elongation of Bar

If the bar elongates (i.e., $L^* > L$), the strain ϵ is positive and is called tensile strain. The negative value of strain ϵ is shortening of bar and is referred to as compressive strain. During the thesis extensional strain is considered uniform along the length of member; such type of uniform strain is called axial strain. Hence, axial strain is given by (Hibbeler, 2018:pp.92–93).

$$\epsilon = \epsilon_{avg} = \frac{\Delta L}{L} = \frac{L^* - L}{L} \quad (2.6)$$

The equation for the axial extension can be derived as:

Let us consider a prismatic body is subjected to equal forces applied in opposite direction along the longitudinal direction and origin of coordinates is in centre of cross section along x-axis. Which in diagram can be represented as:

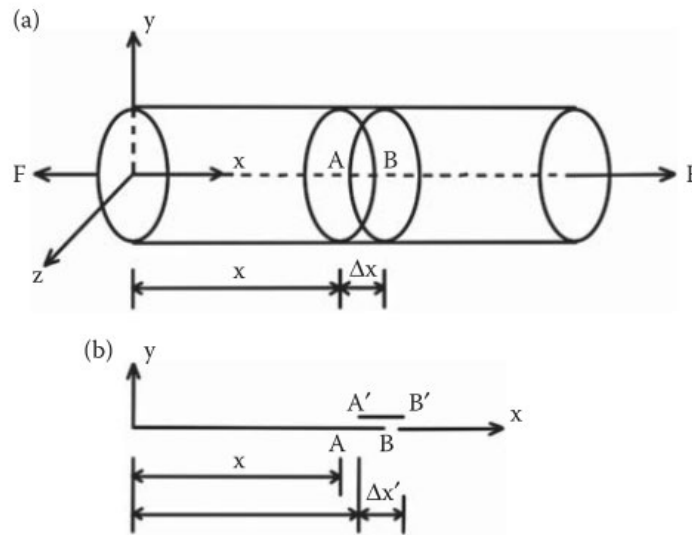


Figure 2 Axially loaded body elongation(Kassir, 2017)

This nature of loading results body to remain in its position and deformation occurs after stretching or shortening of length along x-axis. For Deformation following two conditions are assumed:

- (1) The axis of member remains straight.
- (2) Y-Z plane remains parallel to the cross section.

Mathematically, let us consider two planes separated by distance of Δx as shown in figure 1. When the load is applied, these planes are displaced and length Δx becomes $\Delta x'$. The Conventional axial strain can be written as(Kassir, 2017)

$$\epsilon_x = \lim_{\Delta x \rightarrow 0} \frac{\Delta x' - \Delta x}{\Delta x} \quad (2.7)$$

For any cross section, the axial cross section is constant and may vary with x, but is independent of y and z.

Equilibrium: The external forces are equal and opposite, so the overall equilibrium is satisfied. For any cross section along the horizontal axis, with internal force of magnitude F , the normal stress is given by(Kassir, 2017) :

$$\sigma_x = \frac{F}{A} \quad (2.8)$$

Where A is the area of cross section. The other remaining stress σ_y and σ_z are assumed to vanish throughout the member. Let us consider an element with an area $\Delta A = (\Delta y)(\Delta z)$, then force can be written as(Kassir, 2017) :

$$\Delta F = \sigma_x \Delta A \quad (2.9)$$

Now integration over the cross-sectional area follows as(Kassir, 2017) :

$$F = \iint_A \sigma_x dA = \sigma_x A \quad (2.10)$$

Rearranging the equation (1.4) gives equation (1.2)

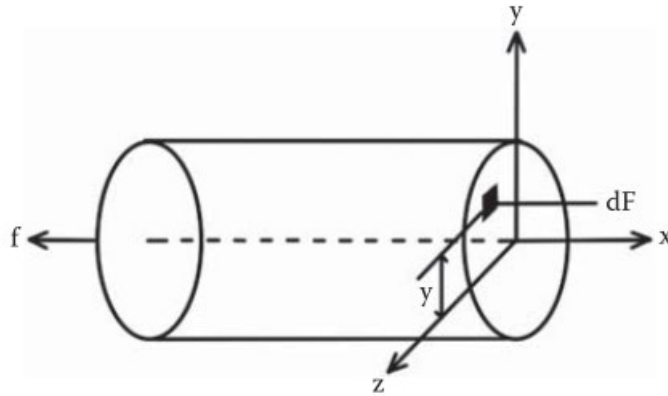


Figure 3 Axially loaded member internal forces(Kassir, 2017)

The moments of ΔF about y and z axes are respectively $\Delta M_y = z\Delta F$ and $\Delta M_z = y\Delta F$.

Hence,

$$M_y = \iint_A z\sigma_x dA = \sigma_x \iint_A z dA \quad (2.11)$$

$$M_z = \iint_A y\sigma_x dA = \sigma_x \iint_A y dA \quad (2.12)$$

The centroid of cross section is origin of the coordinates, so it follows $\iint_A z dA = \iint_A y dA = 0$. From equation (2.11) and (2.12) we can write $M_y = M_z = 0$. The axial force is only the internal force acting on the member.

The elongation or contraction of member is given by (Kassir, 2017)

$$e = \int_0^L (dx' - dx) = \int_0^L \epsilon_x dx \quad (2.13)$$

Where L is the original length of the member .

2.2.3 Young's Modulus(E)

Young's modulus is a numerical constant named after 18th century English physicist Thomas Young, which describes the elastic properties of solid undergoing in only tension or compression. In simple young's modulus is the measure of ability of material to withstand changes in length .This value is only valid for one direction, that is being stretched or compressed lengthwise and returns to its original length after the removal of applied load. Sometimes it is also referred as modulus of elasticity. Mathematically it is equal to the longitudinal stress divided by the strain (Davis, 2004).

$$Young's\ modulu's = \frac{Stress}{Strain} \quad (2.14)$$

Taking the values of stress from equation(2.3) and strain from equation (2.5) and substituting in equation(2.14). we get.

$$E = \frac{\frac{F}{A}}{\frac{\Delta L}{L}} \quad (2.15)$$

Rearranging equation(2.15) gives.

$$E = \frac{F * L}{A * \Delta L} \quad (2.16)$$

Tensile test is performed to calculate stiffness of material using young's modulus. Different materials show higher or lowers young's modulus value. The higher value of young's modulus indicates that the material is excellent for creating strong elements. The SI unit of young's modulus(E, or less commonly used Y) is pascal, but practically most used units are megapascal (MPa) or gigapascals (GPa or $\frac{kN}{mm^2}$) (Davis, 2004).

2.3 Uniaxial Tensile Testing

Tensile testing is performed in selecting the material for engineering application. Mainly when new materials are developed tensile test are performed to ensure the quality and compare with other material. The result obtained from tensile test are used in prediction of mechanical of material (Davis, 2004) .

The standard tensile test was first published and their continuous are now through ASTM and ISO organization. Reliable tensile test data are generated through computer-controlled machines. For Polymeric material tensile test plays a huge important, since they depend strongly on strain rate because of viscoelastic properties. During their application, time dependent deformation like relaxation and creep are shown as higher as compared to metals. Generally multiple temperatures and strain rates are used to fully characterize polymer material(Srinivas, 2017) .

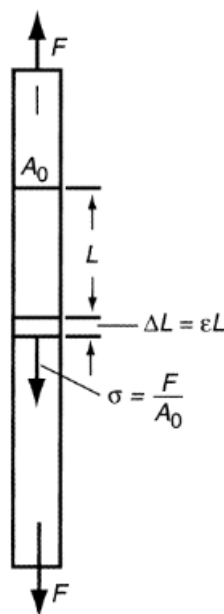


Figure 4 Uniaxial Tensile Test Diagrammatic representation(Davis, 2004)

The above diagram shows uniaxial test on a material The X axis describe the strain and Y axis depicts the strain. The values for stress and strains are calculated as(Srinivas, 2017)
:Stress(σ):

$$\sigma = \frac{\text{Load}}{\text{Area of the material sample}} \quad (2.17)$$

And strain(ϵ)is calculated as:

$$\epsilon = \frac{\text{Change in length}}{\text{Initial length}} \quad (2.18)$$

The slope of initial portion of curve gives the value of Young's Modulus (E).

Mathematically expressed as(Srinivas, 2017) :

$$E = \frac{\sigma_2 - \sigma_1}{\epsilon_2 - \epsilon_1} \quad (2.19)$$

Where:

$\sigma_1 = \text{Intial Stress}$

$\sigma_2 = \text{Final Stress}$

$\epsilon_1 = \text{Intial Strain}$

$\epsilon_2 = \text{Final Strain}$

2.3.1 Importance of uniaxial test

The basic understanding is that uniaxial test is carried out to understand about the ultimate strain and stress of material, information about the factor of safety. The information about factor of safety is foremost while selecting the material. It can used for more application like:

- (1) The result obtained from uniaxial tension tests are used to calculate endurance limits in design calculations.
- (2) Fatigue life of engineering materials.
- (3) By using techniques like Arrhenius equation service life can be predicted.
- (4) In the test procedure aging and other environmental factors can be incorporated to characterize the material.

- (5) In rubber manufacturing industry, it is used to maintain consistency of material and determine the batch quality.

2.4 Linear Elasticity; Hooke's Law

2.4.1 Hooke's Law

Hooke's law or Law of elasticity was discovered by the English scientist Robert Hooke in 1660. For relatively small deformation of an object, the size of deformation or the displacement is directly proportional to deforming force or load. This condition is only valid if the deforming body returns to the original force when the applied force or load is removed. The deforming force may be applied by compressing, stretching, squeezing, bending, or twisting (Anon, n.d.).

Mathematically, Hooke's law states that the applied force F equals a constant (k) time the change in length or displacement (x).

$$F = kx \quad (2.20)$$

The value of k depends not only on the elastic material under condition but also in the shape and dimension (Anon, n.d.).

For uniaxial stress applied to homogenous isotropic member oriented along x axis. The linear relationship between stress and strain is given by below equation and applies for $0 \leq \sigma \leq \sigma_y$. Therefore (Hibbeler, 2018:pp.48–50),

$$\sigma_x = E \varepsilon_x \quad (2.21)$$

Where:

σ_x = Stress along x axis

ε_x = Strain along x axis

E = Modulus of elasticity or Young's modulus

2.4.2 Poisson's Ratio

When a member elongates in axial tension, there is a transverse contraction. This transverse contraction during a tensile test is related to the longitudinal elongation by (Hibbeler, 2018:pp.92–93)

$$\epsilon_{transverse} = -\nu\epsilon_{longit} \quad (2.22)$$

Where ν (Greekl symbol nu) is the poison's ratio. Poisson's ratio is dimensionless quantity with typical values in the range of 0.25- 0.35 range. For simple case of uniaxial stress, it is given as (Hibbeler, 2018:pp.92–93)

$$\epsilon_y = \epsilon_z = -\nu\epsilon_x \quad (2.23)$$

2.5 Stress- strain relations

Stress- strain diagram is the plot of stress versus strain which is used to deduce several significant mechanical properties of materials. All the students of mechanics of material will encounter them often. However, for the case of ductile materials it is totally different. Since they go substantial geometrical change during testing. This part of thesis provides a preliminary discussion of several points needed to understand these curves, and in doing so, it will also provide an introductory overview of several aspects of material mechanical properties. Perhaps the most important test in determining the material properties (Anon, 2021) and done most during the academic year also.

When a specimen is loaded in tensile machine the change in length with increased loading is measured. The data obtained from tensile test, load against extension or stress against strain is recorded and graph are drawn. The typical example of stress strain curve for ductile material is shown as below (Srinivas, 2017):

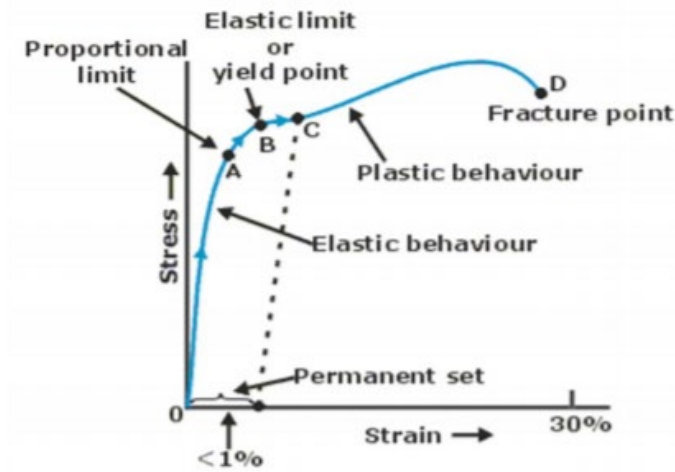


Figure 5 Stress Strain Relationship of ductile material

To analyse the deformation on specimen in elastic plastic region it is compulsory to define stress strain relationship both in elastic and plastic region. A curve obtain from stress strain test is the basis of this relationship. Hooke's Law is the governing equation in elastic region and is defined by (Anon, n.d.).

$$\sigma_x = E \varepsilon_x \quad (2.24)$$

In the early region of curve, many materials obey Hooke's law. As strain increased, they deviate from linear proportionality(Point A). The point of departure is termed as the proportional limit. The nonlinearity is because of stress-induced plastic flow in the specimen. From this position material is undergoing a rearrangement of internal molecular structure, in which atoms are moving towards the new equilibrium positions. Materials lacking this mobility are usually brittle rather than ductile. The stress-strain curve for brittle material is linear over their full range of strain, eventually resulting in failure without appreciable plastic flow (Anon, n.d.).

2.5.1 Elastic Limit or Yield Point

With the slight increase above the proportional limit will result in breakdown of the material and cause it to deform permanently. This behaviour is called yielding. The stress(σ_{yield}) that causes yielding is called yield stress and the point where it occurs is called yield point (Point B). The deformation that occurs is defined as plastic deformation. The upper the yield point, the higher the load carry capacity and vice versa.

Once the material reaches this point it continues to elongate(strain) without any increase in load. This is often referred as being perfectly plastic (Dowling, 2012:p.933).

2.5.2 True strain Vs Engineering Strain

Engineers mostly works with engineering stress. Engineering stress is defined as force divided by the original area of the specimen before loading. once the load is applied then area starts to decrease. True stress is the value of stress in the material lying the actual area of the specimen. The decrement in the area leads true stress being higher than engineering stress (Venkata Deepthi, Sridhar Reddy & Satyadevi, 2018:pp.412–418).

The below figure(a) is graphical representation of engineering stress vs true stress. From figure it is evident that for elastic region the value is same but when the body starts to deform i.e., moving from elastic to plastic region then the value of true stress gradually increases. However, the value of stress in true stress-strain curve always increases because the instantaneous value of area is used when calculating true stress. The reason for this is the reduction in specimen area, outweighs the force reduction. All the design is done typically to operate in linear elastic region. It is uncommon to design product that is supposed to perform beyond the elastic limit (Venkata Deepthi, Sridhar Reddy & Satyadevi, 2018: pp.412–418).

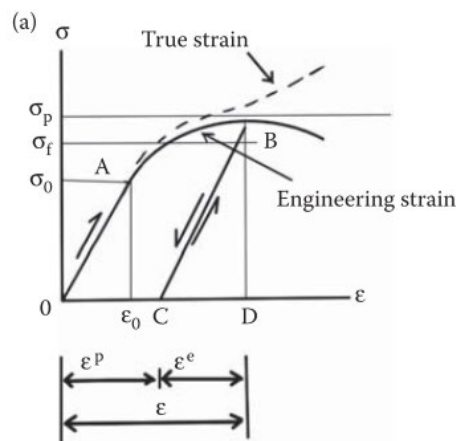


Figure 6 Engineering Strain Vs True Strain

2.6 Elastic Deformation and Plastic Deformation

If the stress is a linear function of strain, then deformation is called Elastic deformation. When the applied force is removed body recovers to original shape and follow Hooke's Law. Plastic deformation is an inelastic deformation that does not disappear when the load is removed. Permanent damaged is caused to material, it occurs when specimen exceed its yield strength and cause it to compress, elongate, bend, buckle, or twist (Pelleg, 2012:vol.190).

Physical reason for the cause of elastic behaviour varies with material and depends on the microscopic structure of the material. For example, the elasticity of rubber and polymer is caused by stretching of polymeric chains under the influence of applied force. Whereas in the case of metal it is caused due reshaping and resizing the crystalline cells of lattices under the direction of mechanically applied load (Venkata Deepthi, Sridhar Reddy & Satyadevi, 2018:pp.412–418; Pelleg, 2012:vol.190).

While determining the elasticity of a material two parameter are studied which are elastic modulus and elastic limit. The high value of elastic modulus suggest that the material is hard to deform. In other words, to achieve a significant strength high load must be applied. The low value of elastic modulus suggest that material easily deforms under the application of load, for example rubber band. The elastic limit is the point on the graph above which the material does not behaves elastically but deforms permanently (Anon, n.d.).

After the writing of Hooke(1676) discussion on elastic phenomena was started. Using continuum approach the real first attempt in constructing theory of elasticity was done . During this only macroscopic phenomenon were described in terms field variables. All the molecular change in structure of body was avoided. After that tremendous amount of effort has been made to define the mathematical theory of elasticity and its physics, engineering applications. It is quite impossible to cover the entire topic only brief introduction to the topic are presented here (Pelleg, 2012:vol.190; Kassir, 2017).

The distance between the atoms (x) is directly proportional to the force they exert on each other. For large distance repulsive force is greater than attractive force and vice versa. The certain distance (x_e) where repulsive and attractive force are zero is denoted as equilibrium atomic spacing. Potential energy at this point is minimum. Strain is ratio of change in x to the x_e .

The interest of elastic deformation in engineering application is usually of small perturbation about the equilibrium spacing which is typically less than 1% of strain. The slope of this small region is approximately constant. If the force is expressed on a unit area on the basis of stress as, $\sigma = F/A$, where F is the applied force and A is cross section of material per atom (Pelleg, 2012:vol.190) .

$$\varepsilon = \frac{x - x_e}{x_e} \quad (2.25)$$

$$\sigma = \frac{F}{A} \quad (2.26)$$

Elastic modulus (E) is the slope of stress-strain relation. We can express it as:

$$E = \left. \frac{d\sigma}{d\varepsilon} \right|_{x=x_e} = \frac{x_e}{A} \left. \frac{dF}{dx} \right|_{x=x_e} \quad (2.27)$$

2.7 Plasticity Theory

The study of plasticity is still on learning. The completed understanding of plasticity is yet not done. It is still active field of learning. The study of plasticity helps engineers to understand mechanism of failures. Hence to design better failure resistant material and structure understanding of plasticity is needed. Sliding of atomic planes over one another , at the molecular level, causes plastic deformation at the continuum scale. When the material is loaded beyond the yield stress plastic deformation occurs (Veerappan, 2017). There are two regimes with respect to with respect to the yield, the first one is elastic regime. It is well known that the loading and unloading curve follows the same path. The second one is the elastoplastic regime. In real materials beyond initial yield. There will be both elastic and plastic deformation. In elastic-plastic regime the loading and unloading path are different. The unloading path is elastic and will be parallel to the elastic line. Let's extend this argument to the strains. Let us say we loaded a material to point A in the elastoplastic regime. Strain at A will be the sum of both elastic and plastic strain (Veerappan, 2017). Mathematically it is expressed as:

$$\varepsilon_{Total} = \varepsilon_e + \varepsilon_p \quad (2.28)$$

The elastic strain can be expressed as stress divided by young's modulus. For plastic strain there is other functional relation given by $F(\sigma)$. Now the total strain can be expressed as

$$\epsilon_{Total} = \frac{\sigma}{E} + F(\sigma) \quad (2.29)$$

Many materials model has been developed to define the functional relation between the stress and strain in plastic region. One of them is Ramberg's Osgood material model. Which defines the functional relation as (Jr, n.d.):

$$\epsilon_p = F(\sigma) = \left(\frac{\sigma}{k}\right)^{\frac{1}{n}} \quad (2.30)$$

Where:

$k = \text{Strength Coefficient}$

$n = \text{Strain hardening exponent}$

On substituting the value of plastic strain in the above equation. The above equation can be rewritten as:

$$\epsilon_{Total} = \frac{\sigma}{E} + \left(\frac{\sigma}{k}\right)^{\frac{1}{n}} \quad (2.31)$$

The above-mentioned equation is called as Additive decomposition of strain and applicable for only small deformation. With regards to stress there is only one stress, and nothing called a "plastic stress". Let's consider points A B and C along the unloading path. If the strain is decomposed, it is found that all the points have same plastic strains and different stresses. This concludes that stresses aren't directly related to plastic strains (Veerappan, 2017).

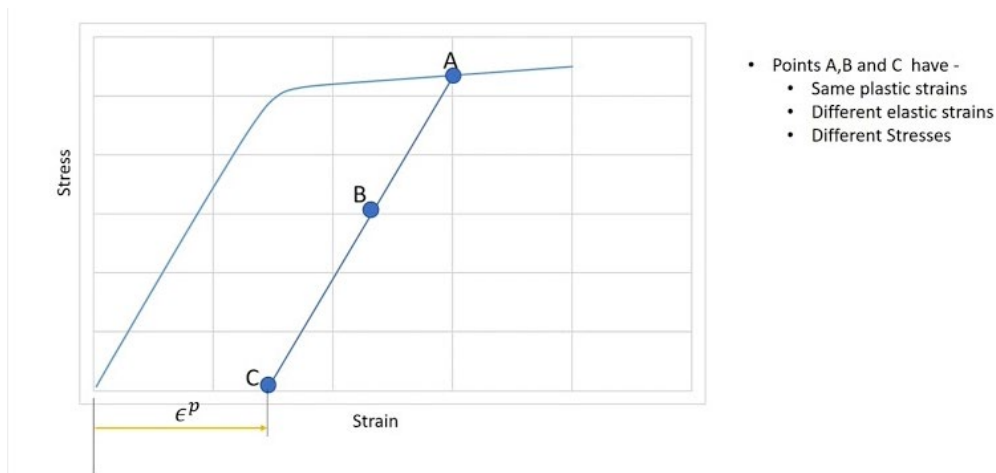


Figure 7 Plasticity modelling(Veerappan,2017)

There are three elements of plasticity modelling. They are:

- (1) Yield Condition**
- (2) Flow Rule**
- (3) Hardening rule**

2.8 Finite Element Method(FEM)

Finite element method of analysis is a very powerful modern of computational tool used almost universally during past 20 years to solve complex structural engineering problems. The first use was particularly in aircraft industry but now it is used in thermal analysis , fluid mechanics, electromagnetics (Logan, 2011).

The process involved is :

(1) Discretize and select the element types.

The first step is to discretize the body into equivalent system of finite element. The associated nodes are selected and appropriate element types to model closely the actual physical behaviour. The total number of elements varies with the shape and size of the body. Selecting the number and size of elements are primarily engineering concern. While selecting the element size it is always good idea to take small element to get useable result yet large enough to reduce the computational effort. In computational software this is done with mesh generation programs (Logan, 2011).

(2) Select a displacement function.

The second step is to select the displacement function within each element. The function is defined using the nodal values of element within the element. Linear, quadratic, and cubic polynomials are frequently used function. Since they are simple to work with finite element method. However trigonometric series can also be used for a two-dimensional problem (Logan, 2011).

(3) Define the Strain/Displacement and Stress/Strain relationships

Strain/displacement and stress/strain relationship are basics for deriving the equation for each element.

(4) Derive the Element Stiffness matrix and Equations.

The development of element stiffness matrices and element equation was based on the concept of stiffness influence coefficients. This presupposes a background in structural analysis using finite element method. Nowadays we have alternative methods which do not require special background (Logan, 2011).

- (4,1) *Direct Equilibrium Method*
- (4,2) *Work or Energy Method*
- (4,3) *Methods of weighted Residuals*

Using any of the above outlined method will produce a set of governing equation describing the behaviour of element. These equations can be conveniently written in matrix form as (Logan, 2011)

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ \vdots \\ \vdots \\ f_n \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & \dots & k_{1n} \\ k_{21} & k_{22} & k_{23} & \dots & k_{2n} \\ k_{31} & k_{32} & k_{33} & \dots & k_{3n} \\ \vdots & \vdots & \vdots & \dots & \vdots \\ \vdots & \vdots & \vdots & \dots & \vdots \\ k_{n1} & \dots & \dots & \dots & k_{nm} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \\ d_6 \end{Bmatrix} \quad (2.32)$$

Or in compact form as:

$$\{f\} = [k]\{d\} \quad (2.33)$$

Where

$\{f\}$ = the vector of element nodal force

$[k]$ = the element stiffness matrix

$\{d\}$ = generalized displacement

(5) Assemble the element equation to obtain the global or total equations and introduce boundary conditions.

In this step the individual element nodal equilibrium is assembled into the global nodal equilibrium equations. The final global equation can be conveniently written in matrix form as (Logan, 2011)

$$\{F\} = [K]\{d\} \quad (2.34)$$

Where

$\{F\}$ = the vector of global nodal force

$[K]$ = the total stiffness matrix

$\{d\}$ = generalized displacement

(6) Solve for unknown degrees of freedom.

(7) Solve for element strains and stress.

(8) Interpret the results.

2.8.1 Linear vs Nonlinear

The generic equation is same for both cases (Logan, 2011).

$$\{F\} = [K]\{d\} \quad (2.35)$$

But for linear the stiffness matrix $[K]$ is constant whereas varying for nonlinear materials.

The graphical representative of the linear and nonlinear graph is as follow:

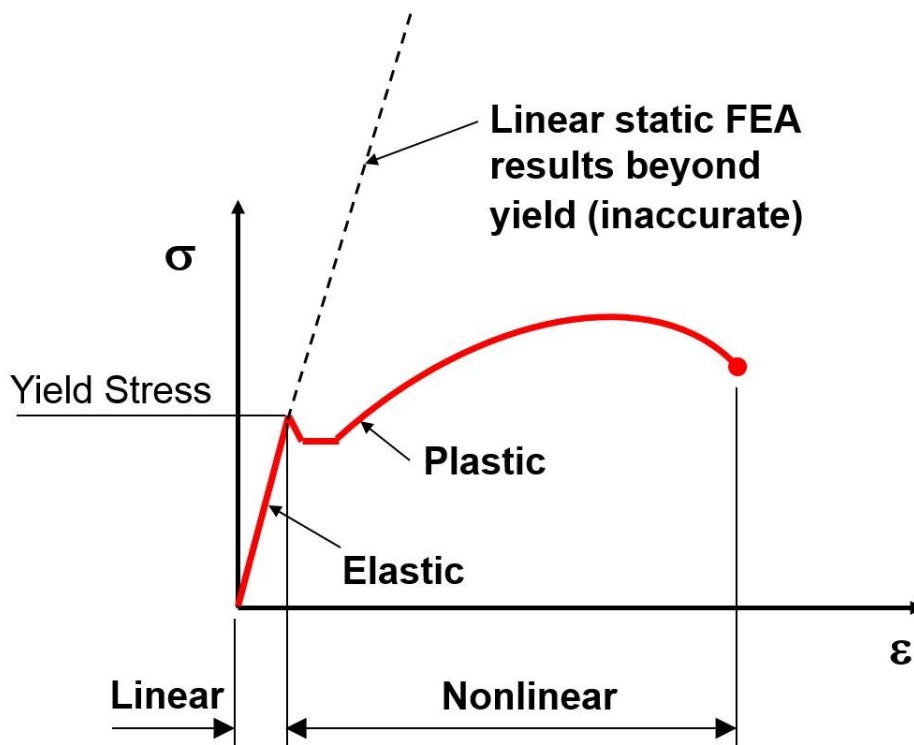


Figure 8 Linear and Nonlinear graph

2.9 COMSOL

COMSOL is a finite element- based modelling software. The tools are well-developed with GUI and several other modules for modelling complex and common types of physics involved in applied science practices and engineering world. In the beginning this software was name as FEMLAB and was written using MATLABTM. The newer version is more user interface with stronger, smarter solver and stands alone package. The latest version is COMSOL 5 series with name 5.5. The addition of Application Builder was Revolutionary. This tool allows users to build application based on model. Meshing is

almost seamless and automatic with a lot of option for structured and unstructured Geometry. The most important feature of COMSOL is that it has an ability to solve any PDE/ODE which may not fit in classical governing equation. The latest Livelink feature allows user to incorporate the software with many available CAD software's(Danielsson, n.d.:pp.3–5) .

In COMSOL setting up and solving the simulation follows one simple standard process. The process is same and common for any simulation. The process involved in modelling in the software is as follows(Danielsson, n.d.) :

- (1) Start by selecting the space dimension.
- (2) Now, add one or more physics interfaces.
- (3) Select the study type that represents the solver or sets of solvers that will be used for computation.
- (4) Provide the necessary variables and condition for the problem.
- (5) Select the material type.
- (6) Select the Mesh Size.
- (7) Run the simulation.
- (8) Calculate the result and plot, export if needed.

2.10 Non-Linear Elasticity Model

The material model that can also describes the phenomena after the yield point are called non-linear elasticity model. In COMSOL the user has ability to select from the inbuilt model. For nonlinear elasticity, the available models are (Gonzalez, 2015)

- (1) Ramberg-Osgood
- (2) Power Law
- (3) Uniaxial Data
- (4) Shear Data
- (5) Bilinear Elastic
- (6) Hyperbolic Law
- (7) Hardin-Dmevich
- (8) Duncan-Chang
- (9) Duncan-Selig

Along with these models the user has an ability to define his own model under used defined option. For this thesis Ramberg-Osgood and Uniaxial is taken.

2.10.1 Advantages

- (1) The foremost advantage is the model is close representative of real life.
- (2) For analysing hyperplastic nonlinear analysis is only way of calculation.
- (3) Dynamic / High speed analysis.

2.10.2 Disadvantages

- (1) Computation time is longer as compared to the linear method.
- (2) The addition increases the complexity to the problem.
- (3) Less stable since almost everything things runs using linear model.

2.10.3 When to use material non-linearity

- (1) When yielding is likely to occur
- (2) For large deformation / Displacement
- (3) Thins parts which may undergo large extension
- (4) High speed loading
- (5) Hyper-elastic / Plastic material (Gonzalez, 2015)

3 REVIEW OF METHODS

The schematic flow process of the methods used is summarized in the below diagram. The process starts with formulating the engineering problem then developing governing mathematical model. Finite element method and analytical method is used to solve numerically. The results obtained from these two methods are verified from the experimental method(Tensile test). All the graphs are plotted, and then conclusion is made from the results.

The first step discussion will be in the values of yield stress since we adopted uniaxial tensile, that is, we are conducting the research with already known values and these

known values will serve as reference for the further steps of this research. Afterwards, the computational values of uniaxial test and experimental are compared.

The algorithms of process in a nutshell are described in below figure:

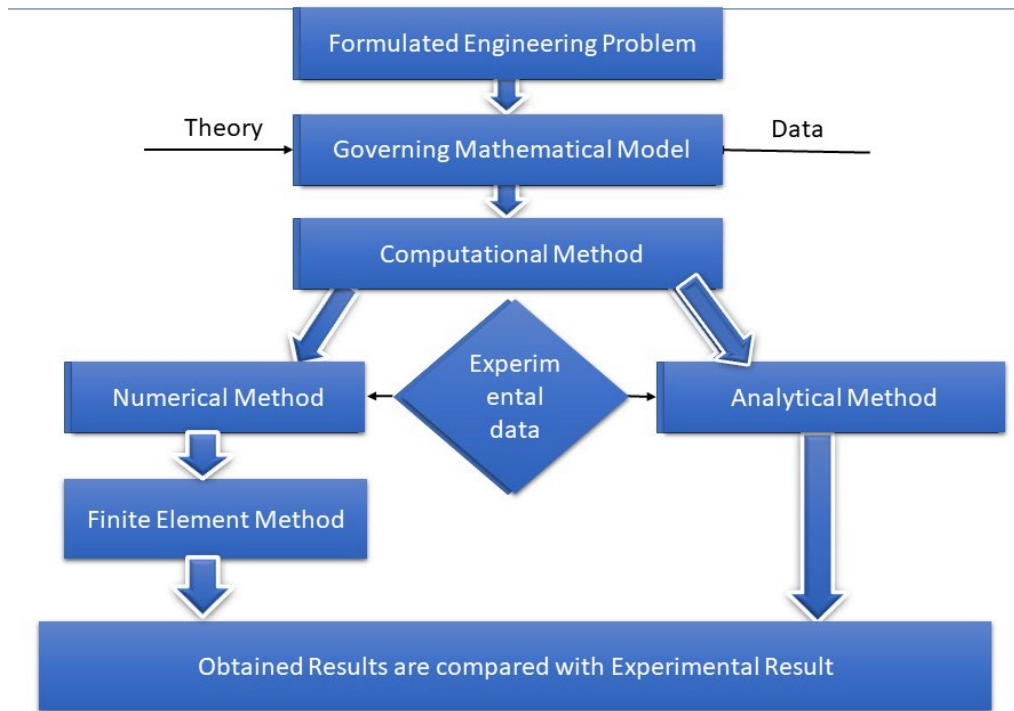


Figure 9 Algorithm of process

3.1 Analytical Methods

For analytical method Hooke's law is used, which in linear region from the basic relation between tension and deformation. The result obtained are plotted and graph is drawn. The linear region serves as the basis for elastic region analysis.

The relation between stress and strain can be described by the Equation :

$$\sigma_x = E \epsilon_x \quad (3)$$

The Equation (2.5) can describe the specific deformation:

$$\epsilon = \epsilon_{avg} = \frac{\Delta L}{L} = \frac{L^* - L}{L} \quad (3.1)$$

We also know that the tension can be described as shown by equation(2.3):

$$\sigma_x = \frac{F}{A} \quad (3.2)$$

Correlating all above equation, we can describe the total deformation as:

$$E = \frac{F * L}{A * \Delta L} \quad (3.3)$$

The (3.3) can be re-written as:

$$\Delta L = \frac{F * L}{A * E} \quad (3.4)$$

In numerical method, despite analysing the section we assume that section is infinite, and the values obtained are absolute consequently committing the mistake disregarding the behaviour of model in other sections.

3.2 Finite Element Method using Numerical approach

Before calculating the required information are

- (1) Material data
- (2) Geometrical data
- (3) External force information
- (4) Boundary condition

Without this above-mentioned information it is impossible to solve the problem. So always before solving the problem these values should be clear. The objective of solving should be known clearly. Generally, calculation is made to find

- (1) Stress
- (2) Strain
- (3) Reaction Force

Let us consider one dimensional element as shown in below figure with material properties

Youngs modulus(E) = E

Length (L) = L

Area (A) = A and fixed at the one end . The horizontal force is applied in the other end.

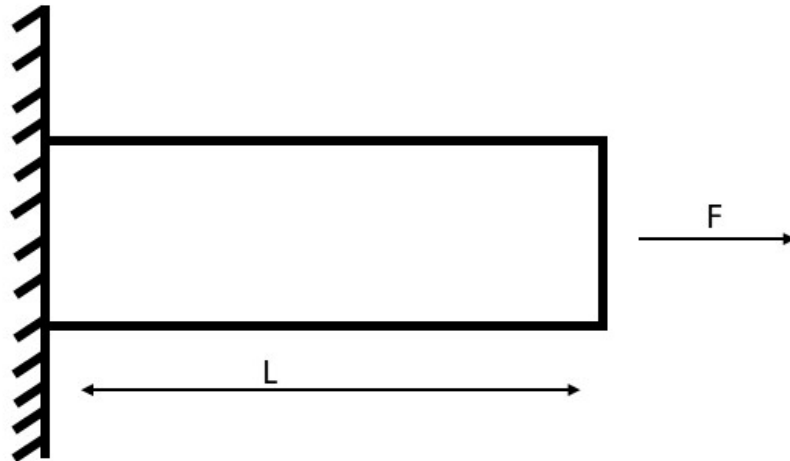


Figure 10 One dimensional bar

For solving the above problem the steps followed are as follows

(1) **Discretization.**

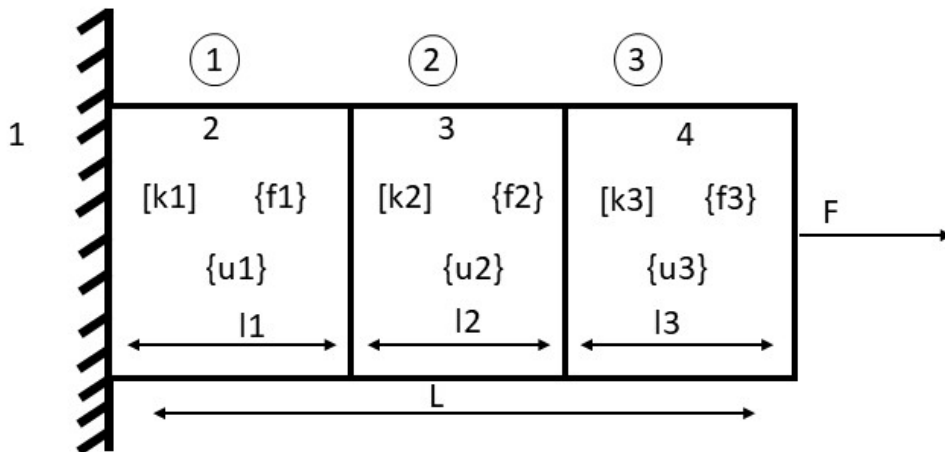


Figure 11 Finite discretization of one-dimensional bar

The bar is divided into three equal parts. While discretizing the body the external boundary condition and external force should go inside with a particular node. There are two types of numbering. The first one with circle and the next without. The number in circle represents element number whereas the normal numbering represents nodal number. The nodal number should always be consecutive.

(2) **Formation of Element-stiffness matrix.**

The stiffness matrix K for bar element in 1 D is given by

$$K = \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad (3.5)$$

For element 1 we must derive $[k_1]$ and similarly for $[k_2]$ and $[k_3]$

(3) Formulation of Global Stiffness matrix.

Once we have found the local stiffness matrix. Then we must assemble them in global stiffness matrix

$$[K_g] = [k_1] + [k_2] + [k_3] \quad (3.6)$$

The size of global matrix is always equal to the number of elements multiplied by degree of freedom. For the above-mentioned case, the number of elements is 3 and the body is assumed to be in 1 D so the size of global stiffness matrix will be 3*3 represented as:

$$[K_g] = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} \quad (3.7)$$

(4) Formation of Global Load Vector.

Similarly, as above we need to find the local force vector and assemble them in global load vector

$$\{F_g\} = \{f_1\} + \{f_2\} + \{f_3\} \quad (3.8)$$

$$\{F_g\} = \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \end{Bmatrix} \quad (3.9)$$

(5) Formation of Global Nodal Displacement.

Similarly, as above we need to find the local displacement vector and assemble them in global nodal displacement.

$$\{U_g\} = \{u_1\} + \{u_2\} + \{u_3\} \quad (3.10)$$

$$\{U_g\} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \quad (3.11)$$

(6) Assemble of Global Nodal Displacement equation.

The global equation is

$$[k]\{U\} = \{F\} \quad (3.12)$$

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} * \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \end{Bmatrix} \quad (3.13)$$

(7) Incorporation of specified Boundary condition.

The implication of boundary condition always results the matrix to smaller form. In the above condition one end is fixed i.e., $u_1 = 0$. The global matrix changes to

$$\begin{Bmatrix} f_2 \\ f_3 \end{Bmatrix} = \begin{bmatrix} k_{21} & k_{22} \\ k_{31} & k_{32} \end{bmatrix} * \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} \quad (3.14)$$

(8) Solution of Simultaneous Algebraic equation.

The linear solution for above problem is as follows:

$$f_2 = k_{21} * u_2 + k_{22} * u_3 \quad (3.15)$$

$$f_3 = k_{31} * u_2 + k_{32} * u_3 \quad (3.16)$$

(9) Computational of Elemental Stress Strain.

In the above equation known values are substituted and unknown values are calculated.

3.3 Finite Element Methods using Software's

From the name it already defines the analysing the model in finite form, all the sections are available for the analysed model. There is an analogy to the analytical data that analogy will result in significant divergence from the simulated over calculated ones.

While analysing the simulation the first consideration should always be the variables and boundary conditions we want to act on the analysed bodies. This is done not only to simulate but accept the values obtained. The test specimen is considered as a global analysis for the analysis of both stresses and total displacement. This is done to obtain erroneous reading of results when compared to experimental and analytical result. The finite element test is performed with the location and magnitude of strain guided by a COMSOL (Finite element model) (Palladino *et al.*, n.d.).

3.3.1 Modelling Geometry through Finite Element method.

For modelling the material test sample requires three governing equations.

- (1) An equilibrium balances.

(2) A constitutive relation relating strain and stress.

(3) A kinematic relation relating stress and strain.

The tensor form of Newton's second law serves as the equilibrium equation. Which is represented as:

$$\nabla \cdot \sigma + F_v = \rho \ddot{u} \quad (3.17)$$

Where

$\sigma = \text{Stress}$

$F_v = \text{Body force per volume}$

$\rho = \text{Density}$

$\ddot{u} = \text{Acceleration}$

For static analysis, the right side of equation goes to zero.

The generalized Hooke's Law governs the constitutive relating the stress tensor (σ) to strain(ϵ) . which is represented as:

$$\sigma = C : \epsilon \quad (3.18)$$

Where

$\sigma = \text{Stress}$

$\epsilon = \text{Strain}$

$C = \text{Fourth - order tensor}$

$:$ = Denotes the tensor product

In COMSOL the relation is expanded to

$$\sigma - \sigma_0 = C : (\epsilon - \epsilon_0 - \epsilon_{inelastic}) \quad (3.19)$$

For this application, the values of initial strain(ϵ_0) , initial stress(σ_0) and inelastic strain($\epsilon_{inelastic}$) are all zero. From our assumption the material is isotropic, the elasticity tensor reduces to the 6 x 6 elasticity matrix:

$$\begin{bmatrix} 2\mu + \lambda & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & 2\mu + \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & 2\mu + \lambda & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix} \quad (3.20)$$

Where

λ and $\mu = \text{Lame' constants}$

Mathematically Lamé's constant is expressed as:

$$\lambda = \frac{2G\nu}{1-2\nu} = \frac{G(E-2G)}{3G-E} = \frac{E\nu}{(1+2\nu)(2-\nu)} \quad (3.21)$$

$G = \text{Shear Modulus}$

$E = \text{Young's Modulus}$

$\nu = \text{Poisson's ratio}$

The kinematic relation between displacement and strains is the final required equation.

Which in tensor form is:

$$\epsilon = \frac{1}{2} [\nabla u + (\nabla u)^T] \quad (3.22)$$

Where T denotes the tensor transpose. For rectangular cartesian coordinates, the indicial notation of the strain tensor may be written as

$$\epsilon_{ij} = \frac{1}{2} \left[\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{\partial u_\alpha}{\partial x_i} \frac{\partial u_\alpha}{\partial x_j} \right] \quad (3.23)$$

Where $\alpha = 1, 2, 3, \dots$. For small deformation, the higher order terms are negligible and ϵ_{ij} reduces Cauchy's infinitesimal strain tensor. The above equation can be expressed as:

$$\epsilon_{ij} = \frac{1}{2} \left[\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right] \quad (3.24)$$

3.4 Calculating the parameters required for modelling

3.4.1 Linear Modelling

Linear Modelling is quite simple the only values needed is Young's Modulus. Although while solving in COMSOL it asks for the Poisson's ratio and Density. The solver can compile the result without these values also.

For calculating the value of young's modulus. In universal testing machine at steady process Tensile test is performed. The obtained are True stress and True Strain. Using these values engineering stress and strain can be calculated. The linear region of graph of engineering strain and stress gives the value of Young's Modulus. Generally, 0,2 % offset method is commonly used.

3.4.2 Non- Linear Modelling with Uniaxial Material Model

The values needed for uniaxial material is stress strain graph which can be easily obtained from tensile test.

3.4.3 Non- Linear Modelling with Ramberg-Osgood Material Model

The generic form of Ramberg- Osgood material model is

$$\epsilon = \epsilon_e + \epsilon_p \quad (3.25)$$

$$\epsilon = \frac{\sigma}{E} + k \left(\frac{\sigma}{\sigma_0} \right)^{1/n} \quad (3.26)$$

Where the strain is sum of elastic and plastic part. The parameters k and n are describing the hardening behaviours of material. The equation used in COMSOL is

$$\epsilon_a = \frac{\sigma_a}{E} + \epsilon_{ref} \left(\frac{\sigma_a}{\sigma_{ref}} \right)^n \quad (3.27)$$

The steps that followed to get the parameters are as follows:

- (1) The stress strain obtained from tensile test are plotted in Excel
- (2) The graph is drawn
- (3) The linear region of graph gives the value of Young's Modulus.
- (4) From graph using 0,2% offset method Yield stress is calculated.
- (5) Using method of nonlinear regression, the value of n is calculated.

4 EXPERIMENT

Testometric tensile testing machine was used for testing the mechanical properties of polypropylene. The machine is connected to win-test analysis software. The material used for testing was SABIC PP 505P. The sample polymer was moulded in injection moulding machine according to ASTM D368 Type 1 Specimen. The dimension of the specimen are as follows:

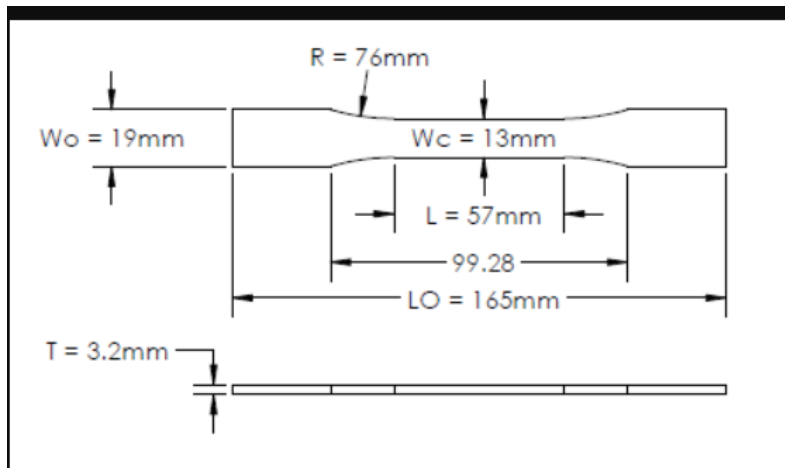


Figure 12 ASTM D368 Type 1 parameters

The software itself is easy to use. The steps that were followed are as follows. The machine was setup to have a prescribed displacement of 20mm/min. Absolute position was fixed at 480,2mm with maximum elongation of 900mm. The length of specimen was setup 165mm and the calculated gauge length was 78mm. The maximum force the machine can exert is 2 ksi.

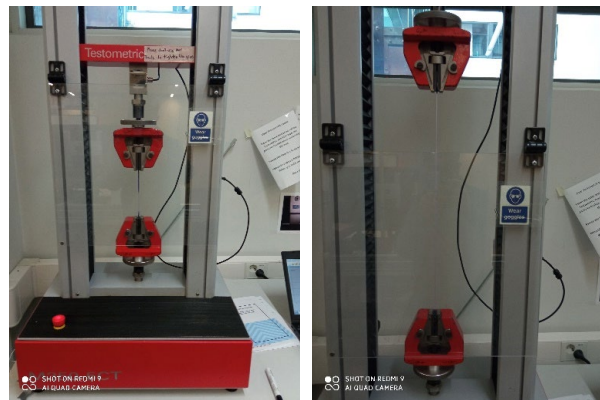


Figure 13 Setup for tensile test (ARCADA B225)

Four different samples were selected. Each sample young's modulus was calculated using in built feature and the output file was .txt with the information on Time, Elongation, Reference value and Force.



Figure 14 Tensile specimen before and after test

The obtained data was processed using EXCEL to calculate the value of Stress vs Strain. The obtained results are as follows:

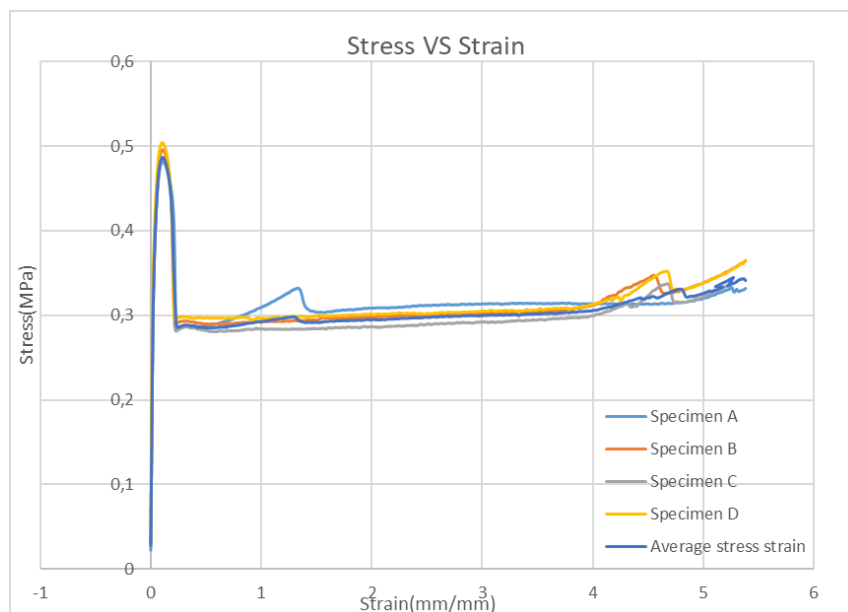


Figure 15 Tensile test (Stress vs Strain) graph

4.1 Poisson's ratio calculation

The calculated value of Poisson's ratio was 0,0253. For the calculation of Poisson's ratio, the ratio of length should be less than one but, in this thesis, it is more so standard value of 0,43 was taken for further calculations.

Abs position	480,2 mm			
Final Disp	900 mm			
Total length	115 mm			
Elongated length	419,8 mm			
Diameter	13 mm			
Average	0,02530368			
Test Specimen	Sample A	Sample B	Sample C	Sample D
Length	87	93	88	80
original length	28	22	27	35
specimen diameter	8,5	8,35	8,34	8,4
Elongated diameter	4,5	4,65	4,66	4,6
Ratio of length	13,99285714	14,20714286	14,02857143	13,74285714
Ratio of diameter	0,346153846	0,357692308	0,358461538	0,353846154
possions ratio	0,024737896	0,025176935	0,025552248	0,025747641

Figure 16 Calculation of Poisson's ratio

5 COMSOL MODELLING

First the geometry was drawn in Solidworks 2020 and imported to COMSOL. While saving the file VRML (*.wrl) format was selected since; in the student version of COMSOL the geometry cannot be directly imported using LiveLink feature.

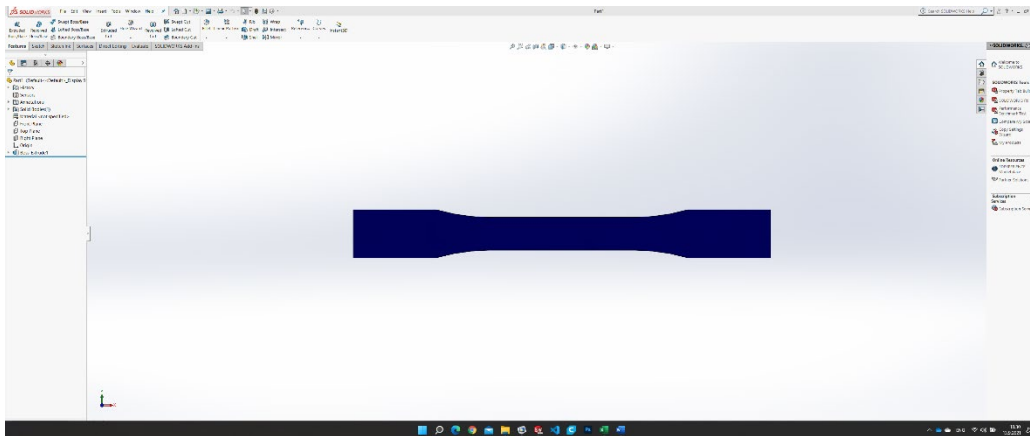
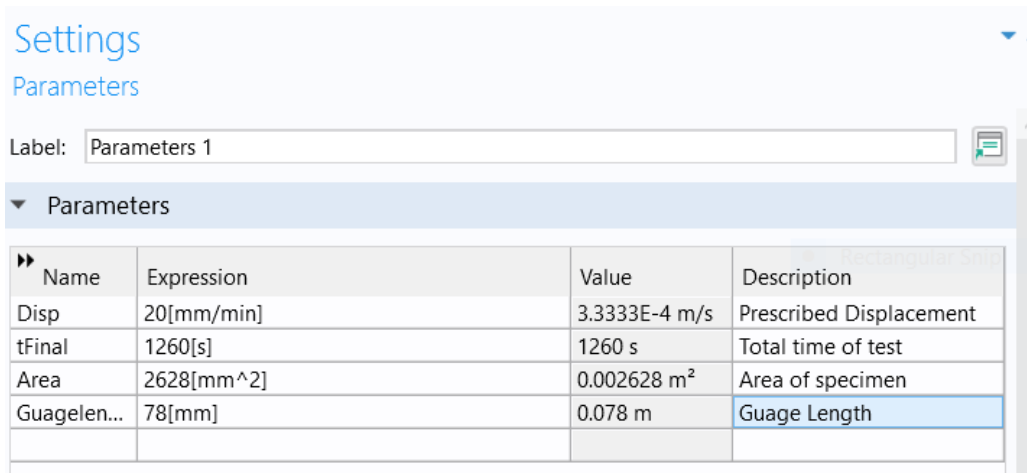


Figure 17 Modelling of Tensile Specimen in Solidworks2020

Under Model Wizard 3D space dimension was selected. Solid Mechanics under structural mechanics module was selected. Since the deformation changes with time so time dependent studies were selected. The used parameters were as follows.



Settings
Parameters

Label: Parameters 1

Name	Expression	Value	Description
Disp	20[mm/min]	3.3333E-4 m/s	Prescribed Displacement
tFinal	1260[s]	1260 s	Total time of test
Area	2628[mm^2]	0.002628 m ²	Area of specimen
Guagelen...	78[mm]	0.078 m	Guage Length

Figure 18 Parameters for defining the values in COMSOL

As to uniaxial test one side is fixed. This is done by applying fixed constraint on boundary 1. The other pulling is done by using prescribed displacement on boundary 8. To match the displacement as to test. The displacement is applied as function of time given by $Disp * t$. The total time of test was 1260 sec with a prescribed displacement of 20 [mm/min]. Mathematically ($\frac{20(mm)}{60(s)} * 1260(s)$), At the end, the total displacement is 420[mm] which is the same displacement measured during the experiment.

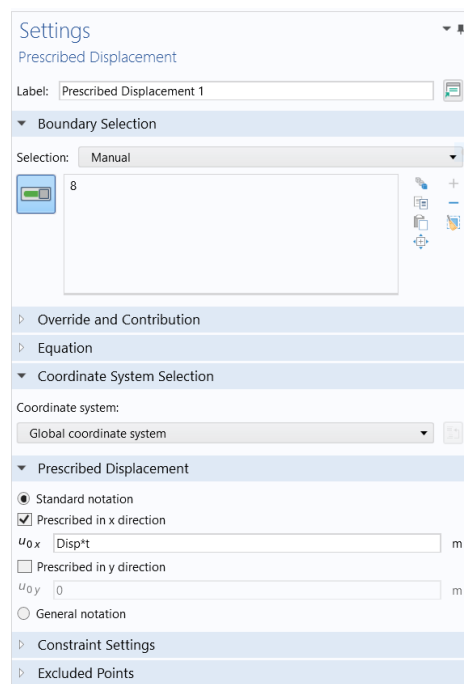


Figure 19 Describing the Displacement

The size of mesh affects the end result. The more finer the mesh the chances of accurate result so extra fine was selected. The configuration of mesh with final meshed geometry is as shown below.

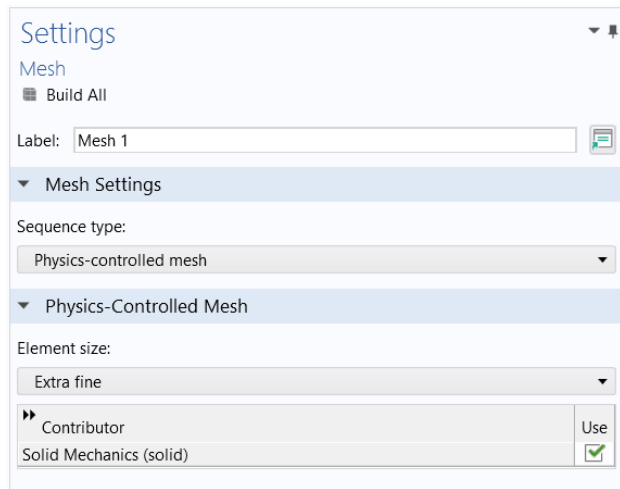


Figure 20 Selection of Mesh

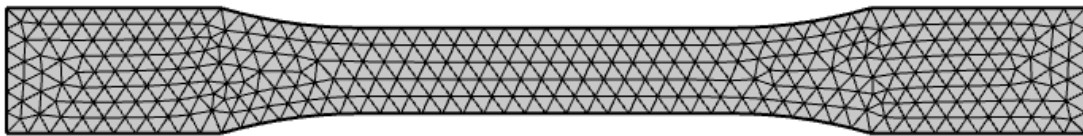


Figure 21 Final Meshed Geometry(COMSOL)

The study was setup as $range(0,0.1,tFinal)$. The time dependent solver will start from zero with an increment of 0,1 and ends at 1260th (tFinal) second.

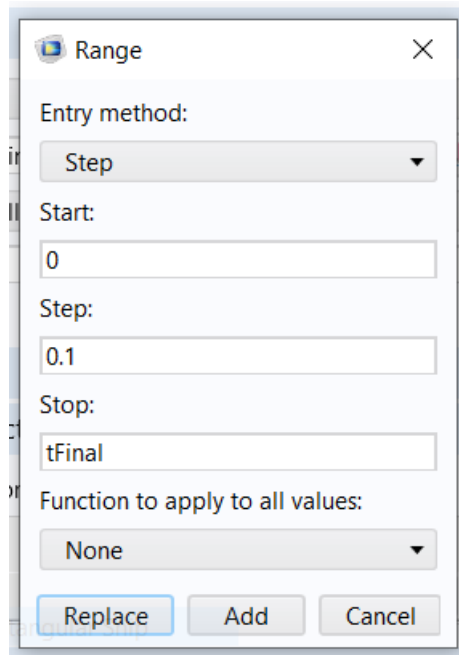


Figure 22 Defining the time boundary

Since behaviour of polypropylene is highly non linear. There were few changes on the default solver. The Relative tolerance was changed from Physics controlled to user controlled with an relative tolerance of 0,001. The time stepping was selected to be fixed and the selected method was BDF. Direct solver was enabled. Under Fully coupled ; Method and temination; Nonlinear method was selceted as *Automatic (Newton)*.

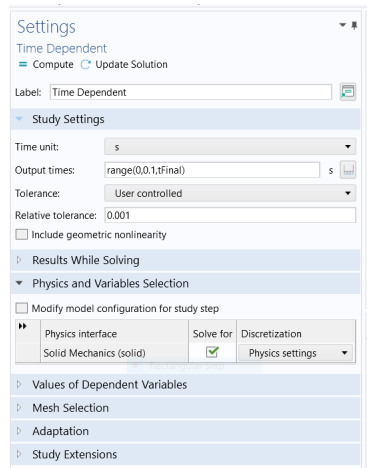


Figure 23 Changing from physics controlled to user controlled relative tolerance

The below first figure demonstrate the enabling of direct solver and second demonstrates selection of Automatic (Newton) method.

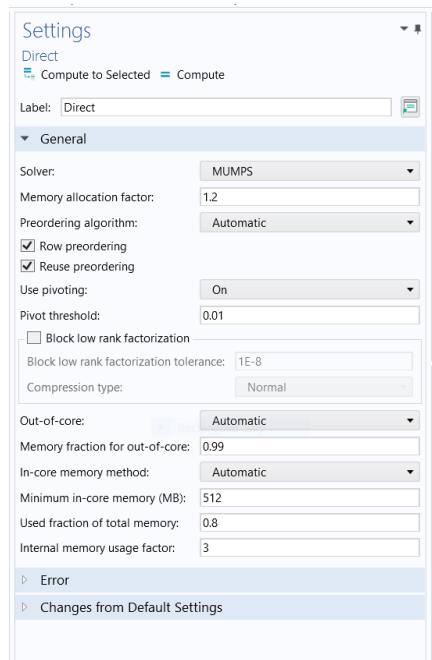


Figure 24 Enabling the Direct Solver

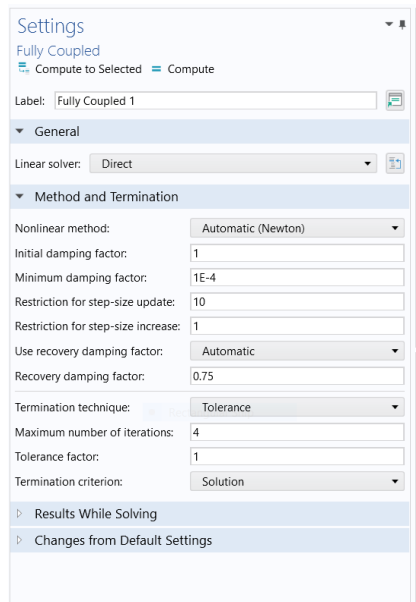


Figure 25 Selecting Automatic(Newton) solver

To obtain the stress strain graph 1D plot was selected. Under one 1D plot point graph was selected. The equation for stress was defined as *stress tensor local coordinate system(Solid.e11)* and for strain; *strain tensor local coordinate system(Solid.e11)* was selected from the dropdown menu of replace extension.

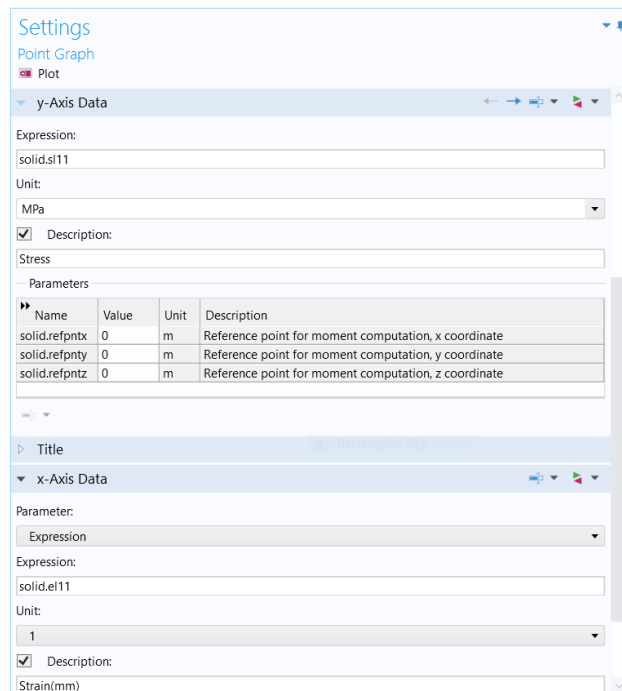


Figure 26 Calculating the Stress Strain values

5.1 UNIAXIAL MATERIAL MODEL

Under solid mechanics module, Nonlinear elastic material model was selected. From the dropdown menu under material model. Uniaxial material model was selected. The stress strain graph was imported as interpolation function.

The first approach in modelling was done by directly importing the stress-strain result from the tensile test. The obtained result couldn't meet the real test. So, the next approach in modelling was done by fitting data. The obtained result and process are shown in the below figure.

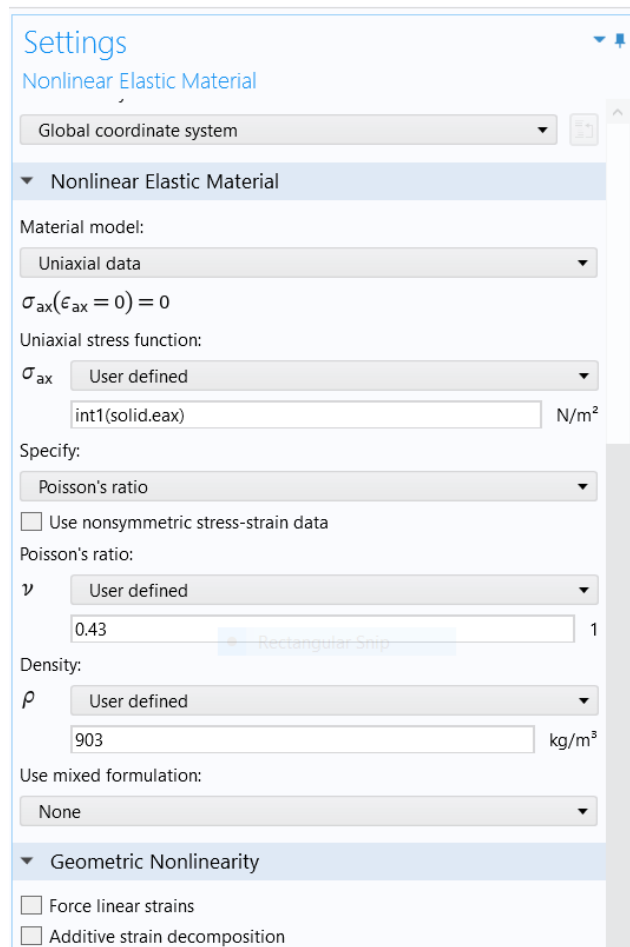


Figure 27 Parameters for uniaxial model

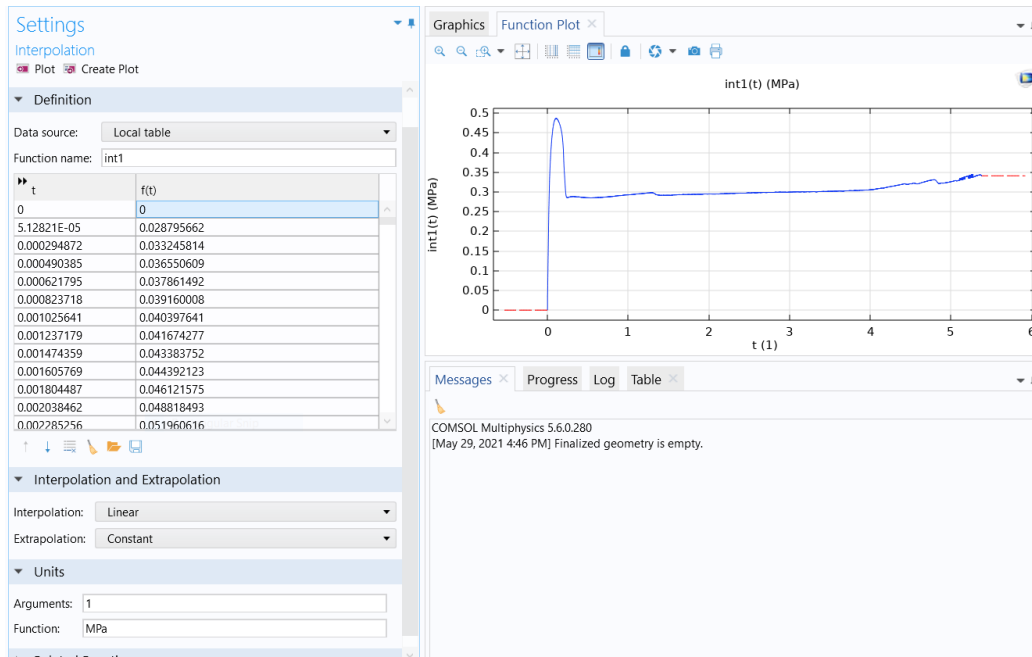


Figure 28 Interpolation curve with real stress strain data

The main problem that aroused using this method, was for any value of displacement the strain developed could not exceed the strain of 0,003. That might because of COMSOL being highly sensitive to data noise. The below graph represents the stress-strain graph obtained using this approach.

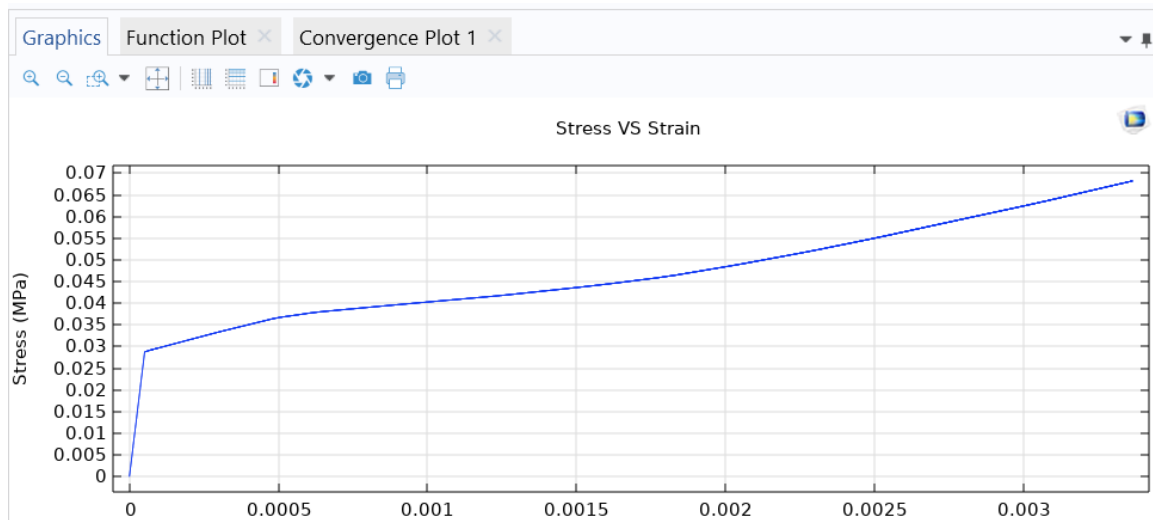


Figure 29 Obtained stress strain graph using real data

So, the second approach was done by reducing the noise from the data. This was done by data fitting with taking the points that could best describe the graph. The graphical representation of fitted data with real data is as shown below.

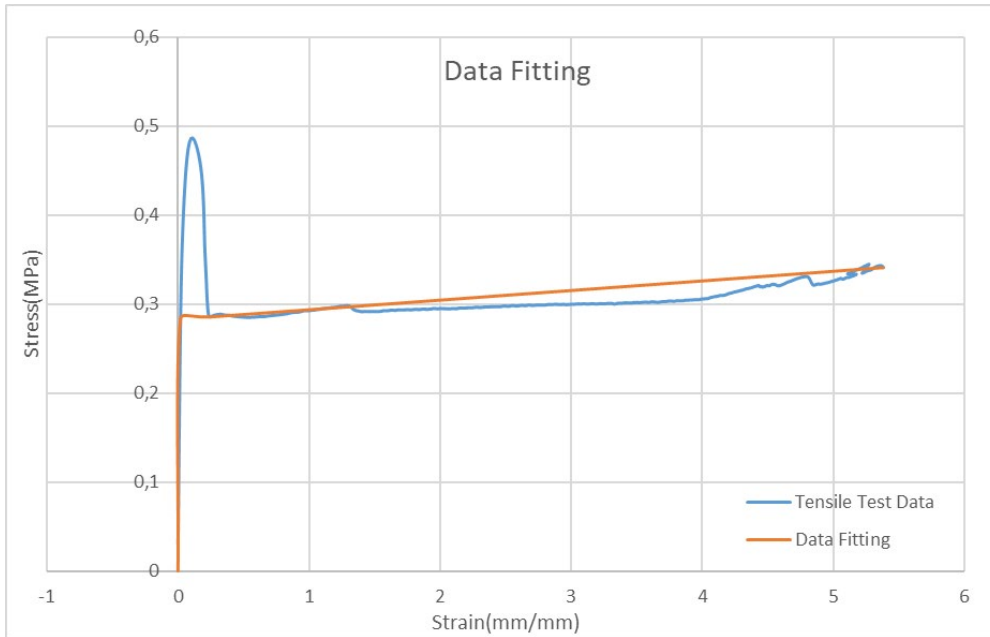


Figure 30 Data fitting for uniaxial model

Again, with the new method the stress strain graph was defined. The test was done again. The below graph shows the interpolation function with fitted data.

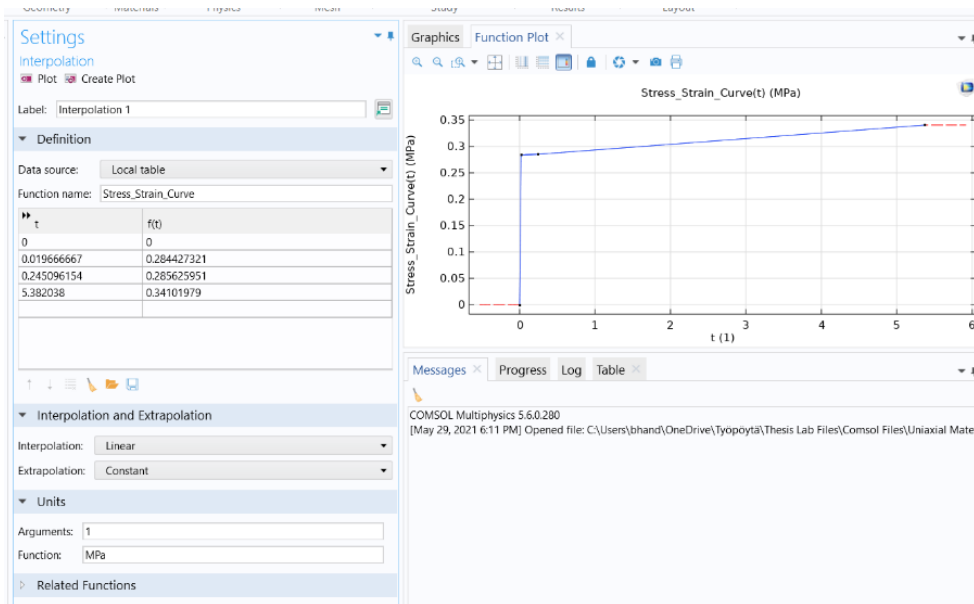


Figure 31 Interpolation curve using fitted data

With the fitted data the obtained stress strain graph is shown in the below figure.

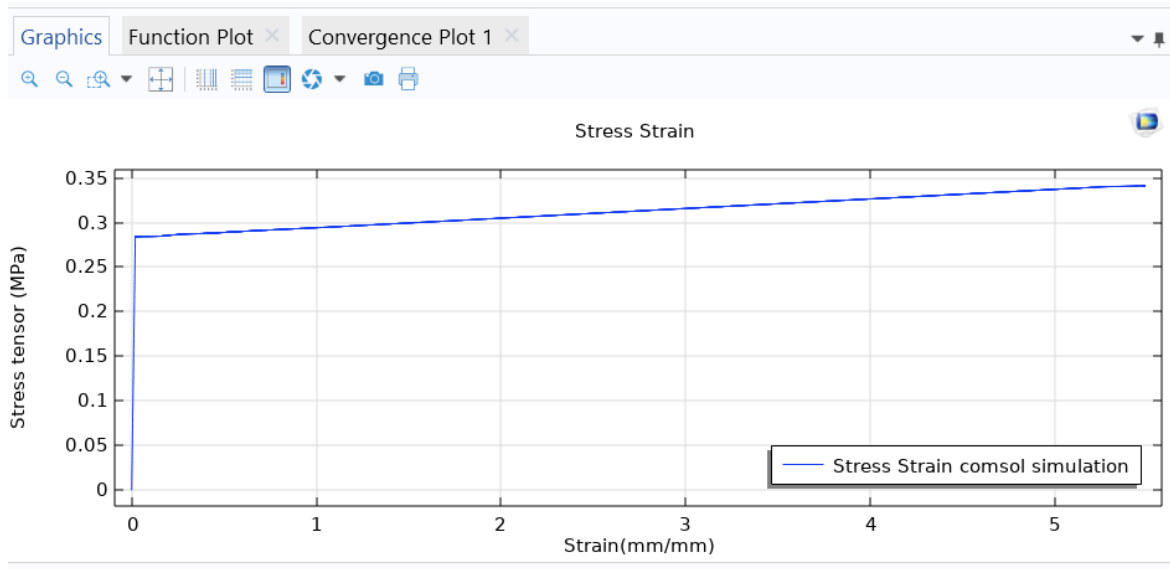


Figure 32 Obtained stress strain graph using fitted data

5.2 RAMBERG OSGOOD MODEL

The first approach before modelling in COMSOL is to find the values of parameter. From the tensile data. The following calculation were made to find the values. The value of Poisson's was calculated above, and same value is used.

5.2.1 Young's modulus calculation

From stress strain graph linear region was taken for the calculation. The calculated value was 14[MPa].

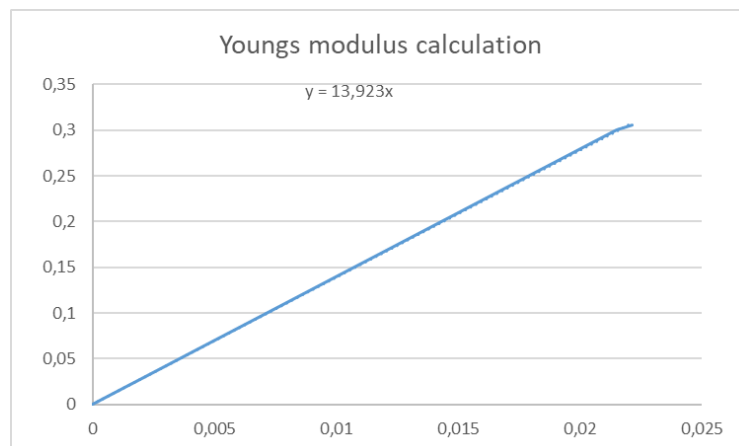


Figure 33 Youngs modulus calculation graph

5.2.2 Reference stress strain calculation

At an offset of 0,2% the reference stress was calculated. The calculated value for stress was 0,302844[MPa] and strain was 0,002.

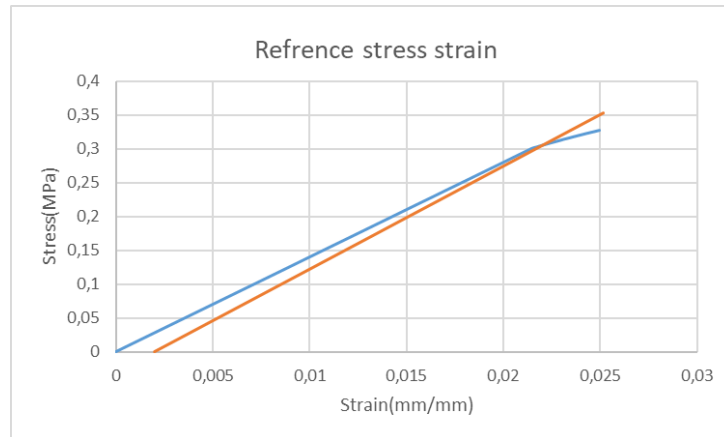


Figure 34 Reference stress strain calculation graph

5.2.3 Stress exponent(n)

The first approach in calculating the stress exponent was done by using nonlinear regression analysis on EXCEL. The nonlinear method in EXCEL failed to find the solution so using hit and trail method the value of n was calculated. The calculated value of n was 190.

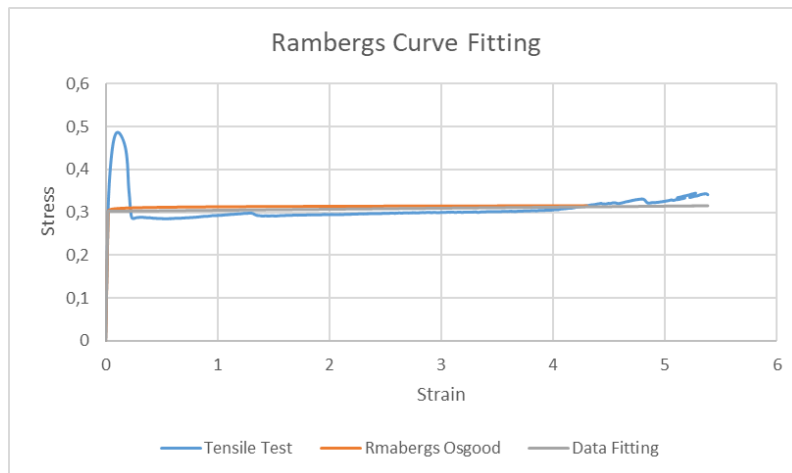


Figure 35 Stress exponent calculation graph

Under solid mechanics module, Nonlinear elastic material model was selected. From the dropdown menu under material model. Ramberg Osgood material model was selected. The input values were as follows.

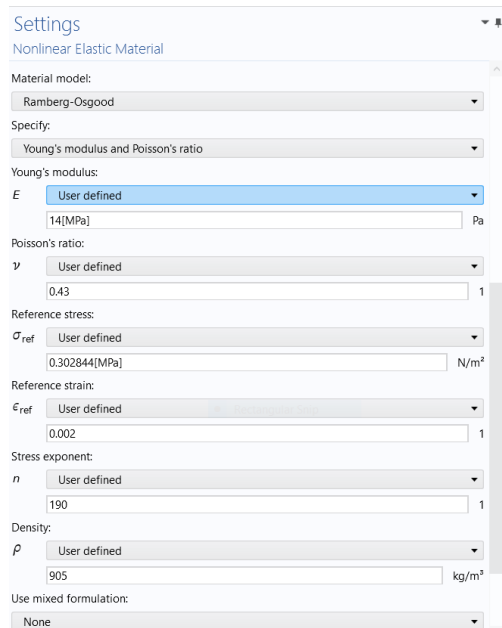


Figure 36 Defining Ramberg's Osgood parameters

Using Ramberg's Osgood material Model, the obtain stress strain was as follow.

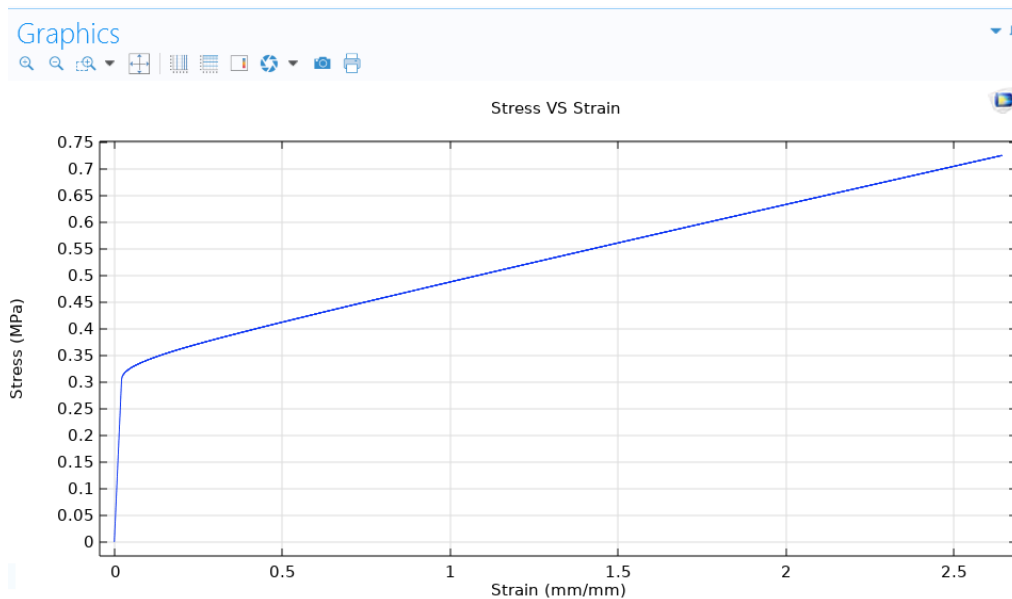


Figure 37 Stress strain graph obtained from Ramberg's Osgood model

The problem that aroused using this method was the strain value of 2,5 which is not true. The conformation was done using stress vs displacement graph.

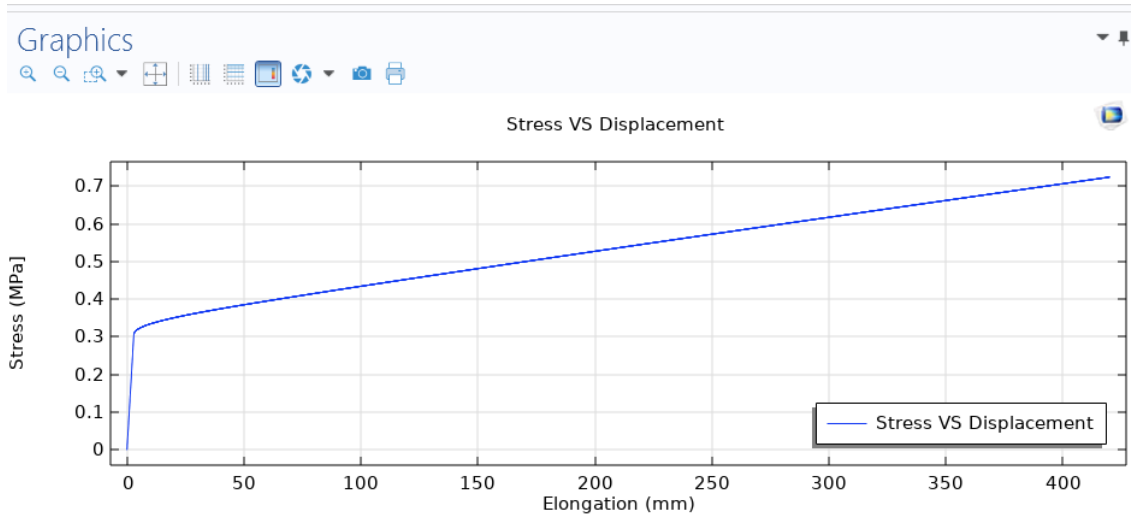


Figure 38 Stress Displacement graph obtained from Ramberg's Osgood model

6 RESULT

6.1 Ramberg's Osgood material model

The obtained result from material model was imported to Excel. The results were plotted along the laboratory results. For Ramberg's Osgood model the displacement value was divided by the gauge length to get the required strain value.

The below figure represents the results obtained from COMSOL and tensile results in a graphical form.

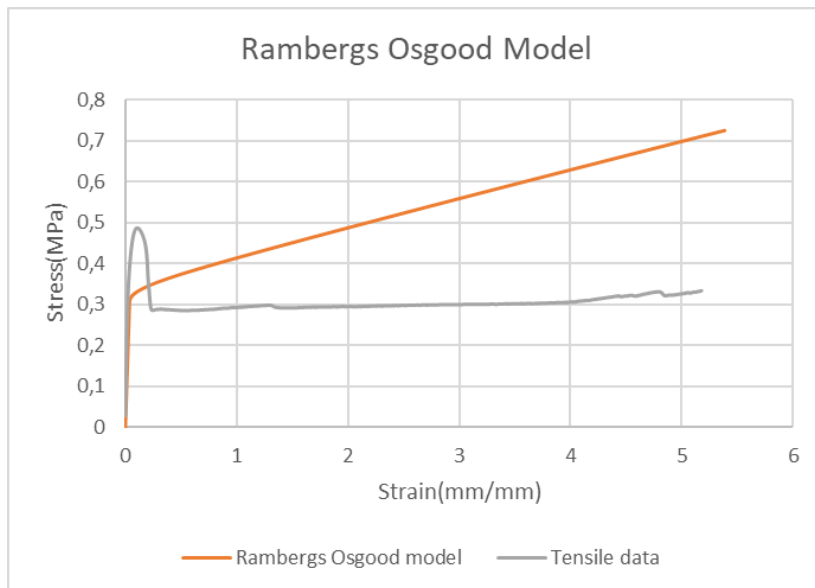


Figure 39 Comparison of Ramberg's Osgood model with tensile result

The obtained stress distribution plot using Ramberg's Osgood model was as follows.

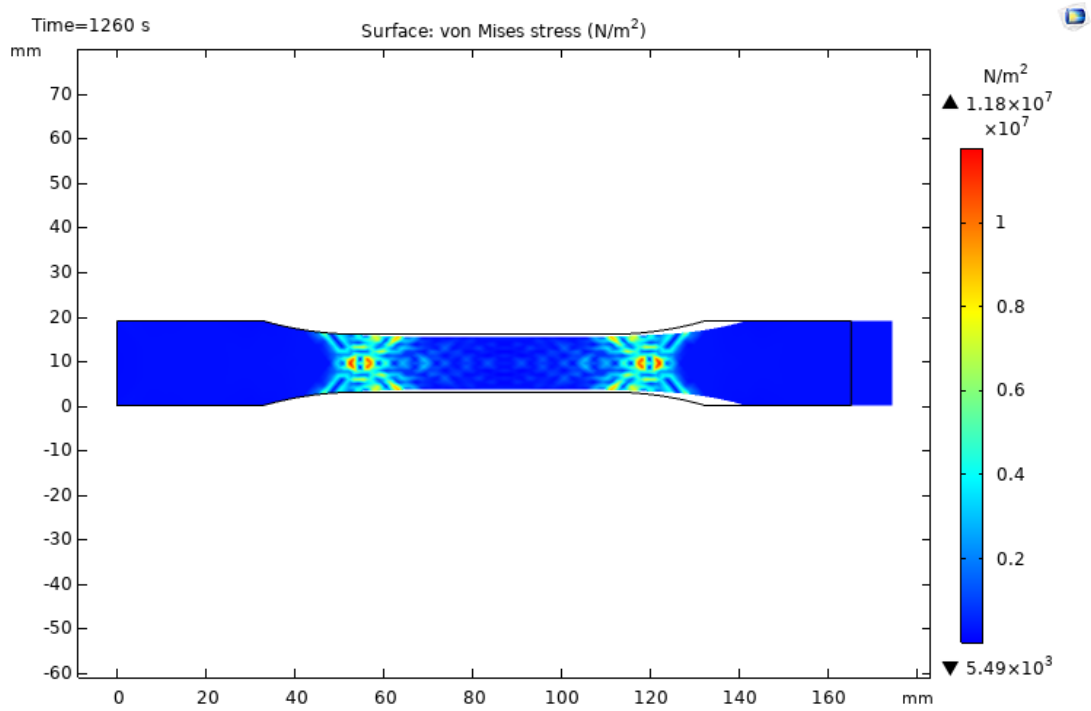


Figure 40 COMSOL stress distribution for Ramberg's Osgood model

6.2 Uniaxial material model

The below figure represents the obtained result from uniaxial material model and tensile result.

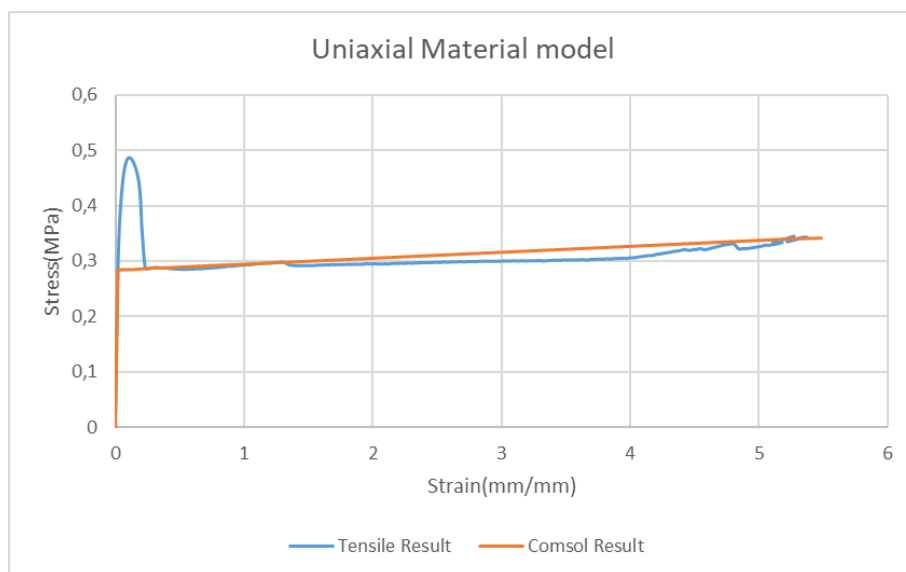


Figure 41 Comparison of COMSOL uniaxial model with tensile result

The obtained stress distribution plot using uniaxial material model is shown below.

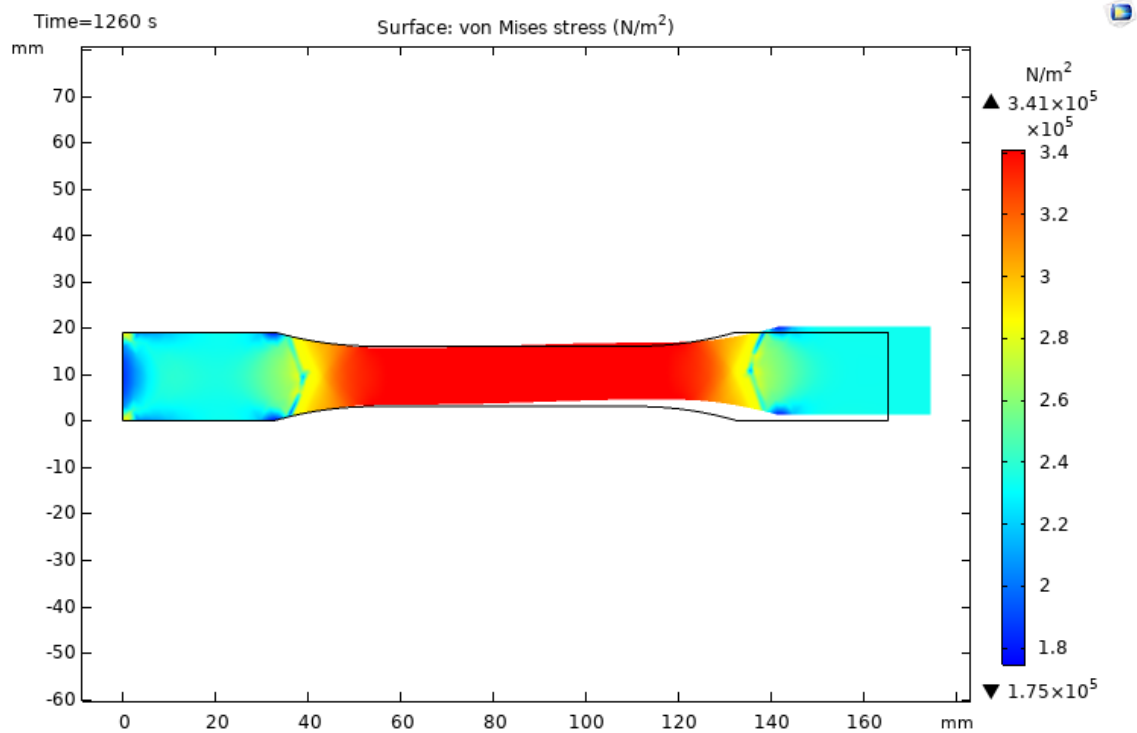


Figure 42 COMSOL stress distribution for uniaxial model

Deviation analysis and error analysis was done to analysed which model could best characterize the nonlinear behaviour. The obtained results were as follow:

Uniaxial Material Model			Rambergs- Osgood Material Model	
Standard Deviation	Stress	Strain	Stress	Strain
Tensile Test	0,032987066	1,58561145	0,032987066	1,58561145
COMSOL	0,023416181	1,681656164	0,116150336	1,554589618
Mean Values	Stress	Strain	Stress	Strain
Tensile Test	0,308505667	2,705260289	0,308504417	2,705157431
Comsol Test	0,312257167	2,763892245	0,532648308	2,692094017
Error Analysis	Stress	Strain	Stress	Strain
Absolute Error	0,0037515	0,058631956	0,224143891	0,013063414
Fractional Error	0,012160232	0,021673314	0,726550021	0,004829077
Percentage error(%)	1,216023193	2,16733142	72,65500207	0,48290772

Figure 43 Deviation and Error analysis

7 DISCUSSION

As stated, modelling of exact behaviour with the available finite element code is challenging task. The experiment was performed using uniaxial material and Ramberg

Osgood material model of COMSOL. According to theory and the properties of polypropylene at low strain rate deformation was expected to rise with time and nonlinear curve was expected to have a linear increment. However, some of the past research and experiment also had a similar result of not having an exact model for modelling the behaviour (Mihai & Goriely, 2017:p.1). So, despite of not having an exact behaviour in nonlinear curve, it is acknowledged as satisfactory to have such result. Still, it is anticipated that the modelling with exact measurement of gauge length to be the possible reason for not having a better result.

The data accumulated from the tensile test was very reliable. The raw or original curve appeared as expected to the nonlinear behaviour of polypropylene. The calculated stress-strain curve deviates from the real data. So, both models couldn't give the exact simulation of phenomena occurred. But on the basis of analysis made from the deviation and error analysis it is assumed that for this experiment uniaxial material model has given the best result. This can be indeed observed from the graph also. It should be noted that for other strain rate Ramberg's Osgood model could as give best result. Since the behaviour of polypropylene changes with the strain rate and the calculation of parameters was limited to EXCEL only. Advance mathematical software's could give best fitted parameters that could change the end results.

8 CONCLUSION

The experiment, calculation and results conducted with polypropylene has resulted a following conclusions:

- The deformation of polypropylene increases with time and nonlinear curve has a linear increment.
- Uniaxial material model being easy in sense, less calculation needed to be done but also give accurate result as compared to Ramberg's Osgood Material model.
- The Nonlinear regression analysis was limited only to EXCEL. Using advance software's like MATLAB to control fitting parameters could give better results.
- The feature of polypropylene is their inelastic behaviour which changes with loading rate effect. For other strain rate another material could serve the best result.

- While simulating in COMSOL temperature was not incorporated. Generally multiple temperatures and strain rates are used to fully characterize polymer material.
- During the thesis extensometer was not used and calculation were done by manual measurement. Using it could get more close result.

9 REFERENCES

Anandarajah, A. (2011) *Computational Methods in Elasticity and Plasticity: Solids and Porous Media*. [Online]. Springer New York. Available from: <https://books.google.fi/books?id=Hsrj1snaYacC>.

Anon (n.d.) *Elastic Deformation and Plastic Deformation*. [Online]. Available from: <https://www.nuclear-power.net/nuclear-engineering/materials-science/material-properties/strength/deformation/> [Accessed: 30 August 2020a].

Anon (n.d.) *Hooke's law | Description & Equation*. [Online]. Available from: <https://www.britannica.com/science/Hookes-law> [Accessed: 7 October 2020b].

Anon (n.d.) *Polypropylene (PP) Plastic: Types, Properties, Uses & Structure Info*. [Online]. Available from: <https://omnexus.specialchem.com/selection-guide/polypropylene-pp-plastic> [Accessed: 30 May 2021c].

Anon (2021) *Stress-strain curve*. [Online]. Available from: https://en.wikipedia.org/w/index.php?title=Stress%E2%80%93strain_curve&oldid=1020659064 [Accessed: 30 May 2021].

Danielsson, M. (n.d.) *How to Implement Elastoplasticity in a Model Using External Materials*. [Online]. Available from: <https://www.comsol.com/blogs/how-to-implement-elastoplasticity-in-a-model-using-external-materials/> [Accessed: 30 August 2020].

Davis, J.R. (2004) *Tensile Testing*. Materials Park, OH 44073-0002, ASM International.

Dowling, N.E. (2012) *Mechanical Behavior of Materials*. Engineering Methods for Deformation, Fracture, and Fatigue. Edinburgh Gate Harlow Essex CM20 2JE England, Pearson Education Limited.

Gonzalez, E. (2015) *Introducing Nonlinear Elastic Materials*. [Online]. Available from: <https://www.comsol.com/blogs/introducing-nonlinear-elastic-materials> [Accessed: 4 January 2021].

Hawkins, R.C. (2007) *Laboratory Turnaround Time*. [Online] 28 (4), 179–194. Available from: <https://www.ncbi.nlm.nih.gov/pmc/articles/PMC2282400/> [Accessed: 30 May 2021].

Hibbeler, R.C. (2018) *Statics mechanics of materials*. [Online]. Lake Street, Upper Saddle River, NJ 07458., Pearson Education. Available from: <http://www.vlebooks.com/vleweb/product/openreader?id=none&isbn=9781292177984&uid=none>.

Jr, G., Ronald J, (n.d.) *On the Ramberg-Osgood Stress-Strain Model and Large Deformations of Cantilever Beams*. [Online]. ScholarWorks@UNO. Available from: <http://scholarworks.uno.edu/td/2377>.

Kassir, M. (2017) *Applied Elasticity and Plasticity*. [Online]. CRC Press. Available from: <https://books.google.fi/books?id=uqU5DwAAQBAJ>.

Koutromanos, I. (2018) *Fundamentals of Finite Element Analysis: Linear Finite Element Analysis*. Linear Finite Element Analysis. The Atrium, Southern Gate, Chichester, West Sussex, PO19 8SQ, UK, John Wiley & Sons Ltd.

Logan, D.L. (2011) *A first course in the finite element method*. [Online]. Stamford, CT, Cengage Learning. Available from: http://bvbr.bib-bvb.de:8991/F?func=service&doc_library=BVB01&local_base=BVB01&doc_number=020342116&sequence=000002&line_number=0001&func_code=DB_RECORDS&service_type=MEDIA.

Mihai, L.A. & Goriely, A. (2017) *How to characterize a nonlinear elastic material? A review on nonlinear constitutive parameters in isotropic finite elasticity*. [Online] 473 (2207), 20170607. Available from: doi:10.1098/rspa.2017.0607 [Accessed: 25 April 2021].

Palladino, J.L., Engineering, D. of, College, T., Hartford, et al. (n.d.) *COMSOL Modeling and Tensile Loading of Aluminum Material Test Samples*.

Pelleg, J. (2012) *Mechanical Properties of Materials*. [Online]. 190. Available from: doi:10.1007/978-94-007-4342-7.

Schröder, J., Wriggers, P., Gross, D., Müller, R., et al. (2017) *Mechanics of materials: formulas and problems : engineering mechanics 2*. [Online]. Berlin, Heidelberg, Springer. Available from: doi:10.1007/978-3-662-53880-7.

Srinivas, K. (2017) *UNIAXIAL TENSION TEST: THE MOTHER OF ALL MECHANICAL TESTS*.

Szpieg, M., Giannadakis, K. & Varna, J. (2011) *Time-dependent Nonlinear Behavior of Recycled Polypropylene in High Tensile Stress Loading*. [Online] 24 (5), 625–652. Available from: doi:10.1177/0892705711399737.

Veerappan, P. (2017) *Basics of plasticity theory*. [Online]. Available from: <https://www.youtube.com/watch?v=LWM9NxOwbA8> [Accessed: 7 June 2021].

Venkata Deepthi, T., Sridhar Reddy, C. & Satyadevi, A. (2018) *Elastic-Plastic Analysis Under Proportional and Non-Proportional Loading Paths Using Deformation Plasticity Theory*. [Online] 19 (6), 412–418. Available from: doi:10.1080/15502287.2018.1534155.