PLASTIC ANALYSIS OF CONCRETE BEAM STRUCTURE USING LUSAS

MOHD AKHMAL BIN MD ARIPIN

A project report submitted in partial fulfillment of the
requirements of the award of the degree of
Master of Engineering (Civil-Structure)

Faculty of Civil Engineering
Universiti Teknologi Malaysia

JUNE 2014

To my beloved parents
Md Aripin Bin Ali
Enah Binti Othman

&

The entire family

All of me loves all of you

ACKNOWLEDGEMENT

“In the Mighty Name of Allah, The Most Beneficent, The Most Merciful”

Firstly, I would like to express my sincere gratitude and love to my dear parents, Mr. Md Aripin Bin Ali and Mdm. Enah Binti Othman and the entire family for the consistent love, continuous support and non –stop guidance that were shown to me during the course of my study

Secondly, my warm and deepest appreciation goes to my supervisor Associate Professor Dr. Suhaimi bin Abu Bakar from whom I received the necessary guidance throughout my project study. With much valuable suggestion contributed lead me to achieving the set aims of the study. Patience however on his part and unfailing support leads to the successful complete of my study. Thank you very much.

Thirdly, I wish to express my thanks to my co-supervisor Dr Mohd Yunus bin Ishak for helping in doing my computing analysis works for sharing valuable data as regards to my study which helped in drawing to a conclusion.

Lastly to all my dear friends whom I am lucky to have, thanks for all your contributions and support, both directly and indirectly towards by project study and the entire degree programmed. We will cherish this moment together. Thank you.

ABSTRACT

Reinforced concrete structure is vital in construction industry and the demand from the users is strictly high and rises each day. Although the usage of concrete structures is being improved day to day, there is always a chance to improve the structure. The limitation of plastic knowledge on concrete beam behaviour has become the reason why there is still disaster that occurred everywhere. Moreover, with the existence of computing software that is capable to complete an analysis in a short time, the usage can be still questionable whether the product is reliable or not. LUSAS was chosen in this study to determine the plastic behaviour of concrete beam model and all the results were compared to experimental data for verification. The concrete beam materials is modelled by concrete (model 94) and reinforcement is modelled by Von Mises Yield Criterion. Based on the analysis, there is a slight difference between LUSAS and experimental data where at the first crack load of the sample, experimental data recorded a deflection of 2.96 mm while LUSAS stated 3.06 mm.The sample failed in flexural with concrete crushing and the results were verified using the stress contour.

ABSTRAK

Struktur konkrit bertetulang merupakan elemen yang penting dalam industri pembinaan. Walaupun penggunaan struktur konkrit semakin berkembang, struktur konkrit masih mampu di tambah baik. Namun, pengetahuan plastik yang terhad ke atas tingkah laku rasuk konkrit merupakah salah satu sebab berlakunya kegagalan ke atas struktur . Selain itu, dengan adanya perisian komputer yang mampu untuk melengkapkan analisis dalam masa yang singkat, penggunaannya masih boleh dipersoalkan sama ada produk itu boleh dipercayai atau tidak . Bagi kajian ini, LUSAS telah dipilih untuk menentukan kelakuan model plastik rasuk konkrit. Untuk tujuan pengesahan data, semua keputusan yang diperolehi daripada analisis LUSAS dibandingkan dengan data ujikaji eksperimen. Rasuk konkrit telah dimodelkan menggunakan konkrit (model 94) dan besi tetulang telah dimodelkan meggunakan Kriteria Alah von Mises. Berdasarkan analisis yang dijalankan, terdapat sedikit perbezaan antara data LUSAS dan data ujikaji eksperimen di mana pada beban retak pertama sampel, data eksperimen mencatatkan pesongan 2.96 mm manakala LUSAS memberikan 3.06 mm. Sampel tersebut telah gagal dalam lenturan dengan penghancuran konkrit dan keputusan telah terbukti dalam kontur tegasan.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOL

a_v/d	-	Shear span to effective depth ratio
σ	-	Stress
ε	-	Strain
δ	-	Deformation

CHAPTER 1

INTRODUCTION

1.1 Introduction

Critical reinforced concrete is needed for the usage of designing a possible total collapse that will lead to a fatal injury to the users. A structure need to be design as a structure that can carry high amount of load while maintaining the efficiency of the structure. The existence of problem such as typhoon or earthquake can make a structure highly dependable on its ductility.

The analysis that should be done to the structure need to be through a laboratory analysis and analytical or theory analysis. Behaviors of reinforced concrete need to be properly define so that the correct reinforcement for concrete can be placed. It is known that properties of concrete can be improved while maintaining its critical characteristic.

Many disaster that involved with engineered structure such as building, had given the human and economy a real problem. Such problems cannot be avoided, thus as an engineer, they have the responsibility in order to reduce the impact came from all the major problems related to engineer. A big number of deaths had been recorded as well as the damaged of the eco-social and the environment.

There are a few ways in order to do research regarding concrete beams. One of the most popular ways is by experimental test. Beams will be casted manually and cured in order to achieve the targeted strength. Meanwhile, finite element is a popular choice of research tool due to the time and cost constraint of the experimental testing. However, the reliability of the finite element model developed must be verified by comparison with the previous laboratory testing results. Finite element can give more flexibility in doing a research since it can test and predict the behavior of a beam that sometimes cannot be tested by the humans. Present study to be carried out with the aims to provide an improved understanding of the behavior of the concrete beams.

1.2 Problem Statement

Concrete beam is the most popular element that is used in a construction of one building. The crucial benefits for concrete are its great compressive strength, which allow the use of bold, slender components and almost unlimited range of forms which is possible to, both in overall appearance and in detail, its variable surface texture; and its range of colours (Heinle, 2000).

However, the behavior of concrete beams is not fully understood and there is a lot more to be investigated. Plastic analysis study is one of the major concerns in any structure design but there is still limitation in the study which leads to not being able to understand the plastic behaviour of a structure. This limitation leads to major or minor problem that is still yet to be overcome by any designer. The study for problem such as plastic behaviour can be analyzed using laboratory experiment and computer software by using non-linear analysis but result for computer software cannot be fully trusted until a verification of the data had been done.

1.3 Objectives

1. To develop the finite element models using LUSAS and analyse based on plastic analysis on the concrete beam
2. To determine the behavior and the stress contour of concrete beam at plastic stage
3. To compare the results from LUSAS and experiment.

1.4 Scope

This research will be fully conducted using a computer software program. A laboratory result will be chosen as a controlled variable which all results, including values and graphs will be compared to the result from the software analysis. The previous experimental testing by Mohd Yunus (2012) was chosen and the concrete will be modeled exactly from the test. Two number of model with the shear span to effective depth (a_v/d) ratio of 5.45 and 3.55 were model based on the available geometry and material properties.

The concrete beam will be modeled using finite element software, LUSAS. LUSAS is finite element analysis software that can analyze all types of complicated models using linear or non-linear analysis. Finite element analysis is the most popular numerical method used in today’s project. Finite element method is one of the methods that analysis a structure or object that is non-easily formulated using ready-based formula.

Computational study was based on displacement and stress contour of concrete beam by using nonlinear finite element analysis. Structural behavior and performance in term of failure mode, cracking or crushing pattern, load-deflection graph of the concrete beam were evaluated.

The research will be focus at the strength and stress of the modeled. The dimension of the beam will be 120 x 250mm and the superposition of nodal degrees of freedom assumes that the concrete and reinforcement are perfectly bonded. The self-weight of the beam is negligible compared with the applied load and that the effects of any shear reinforcement can be ignored. The concrete beams were model in the half size since it is symmetrical.

The graphs of load-displacement were compared to see the length of the ductility on the beam. The method used in this analysis is depending on the capability of LUSAS Software. The stress contours were observed in order to investigate the behavior of failure.

CHAPTER 2

LITERATURE REVIEW

2.1 Finite Element

2.1.1 Introduction to Finite Element

Finite element method is a numerical method to find out an approximate solution for variables in a problem which is difficult to obtain analytically. Finite element method is on the procedure of importantly converting structural problem into a mathematical model which is crucial. Most of the finite element problem need to be properly studied before any model design can be made so that it can be prepared in the most realistic way and give the most accurate value. Results from finite element will not be the same when compared to the experimental data so it need to be interpreted in details before it can be used in other project design (Robson, 2001).

Together with the appropriate coordinates and vectors,the techniques, the model shape, geometry, applied loads, constraints, displacements and stresses have to be defined relative to some reference system, in order to develop and apply the concept of finite element method or systems (Robson,2001).

2.1.2 Basic Theory and Concept of Finite Element

How finite element models are created and human analysis are the dependency on the precision of the analysis. It require a lot of experience or judgment, intuitive thinking on part of those structural analyst in the phase of analysis (Rajasekaran,2003).

The material of a structure undergoing deformations with external loads has characteristic which related to the forces or stresses with displacement or strains. These are ascertained from experiments. Most of the time Hooke’s law provides the basis of linear relationship between force or stress to displacement or strains.

Proper boundary conditions are needed for a structure to resist any external load of its own self-weight (Rajasekaran,2003).

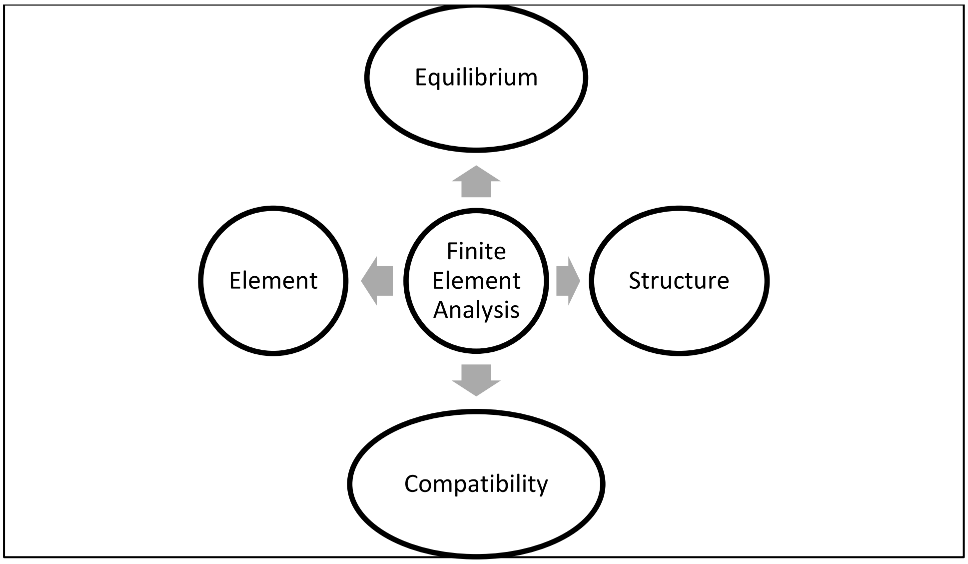


Figure 2.1 Stiffness Approach

2.1.3 Linear Analysis

In linear analysis, it is assumed that the overall structural response is linear, and implies linearity of both the geometric and material response. This type of analysis is only suitable for simple structure, for example, the structure without tall frame and no large rotation.

2.2 Non Linear Finite Element Analysis

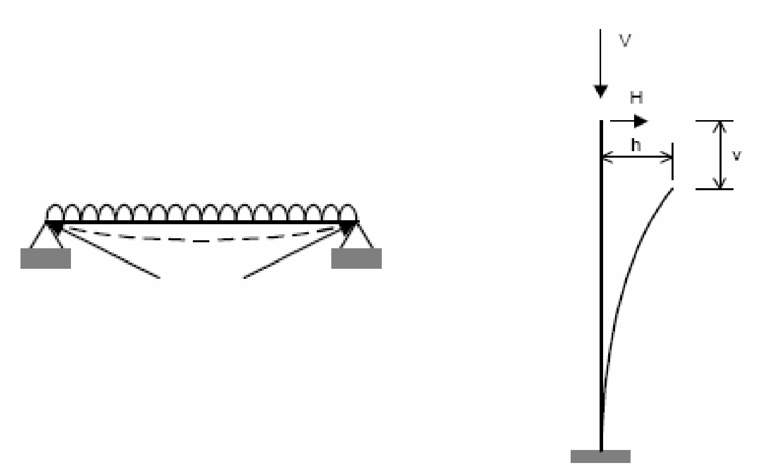
All materials are assumed to have deformations are small enough and linear elastic behavior to significantly affect the overall behavior of the structure in a linear finite element analysis,. However, this analysis is limited to very few situations in the real world, but with a few restrictions and assumptions; linear analysis will be sufficient for the majority of engineering applications (Julien,2003).

However, nonlinear finite element analysis is required in situations such as gross changes in structural geometry, permanent deformations, structural cracks, buckling, stresses greater than the yield stress and contact between component parts. The nonlinear analysis generally can be divided into three types; geometry nonlinearity, boundary nonlinearity and material nonlinearity (Julien,2003).

2.3 Geometric Nonlinearity

Geometric nonlinearity occurs when there is significant change in the structural configurations during loading. Common examples of the geometric nonlinearity are plate structures which developed membrane behaviour, and the geometric split of truss or shell structures. Figure 2.2 shows two simple structures which had geometrically nonlinear behaviour. In Figure 2.2(i), the linear solution would predict the simply supported bending moment and assumes zero axial force in the simply supported beam. But, in reality as the beam deforms its length increases and an axial force will occurred.

For the loaded strut in Figure 2.2(ii), the linear analysis would fail to consider the progressive eccentricity of the vertical load on the bending moment diagram. These examples show the importance of nonlinear geometry in structural analysis.



(i) (ii)

Figure 2.2 Examples of geometry nonlinearity behaviour

2.4 Boundary Nonlinearity

In boundary nonlinearity, the modifications to the external restraints resulting from deformation process such as lift-off, or smooth or frictional contact are taken into account within an analysis. Figure 2.3 shows the structure and its supporting surface which can resist being pushed together, but not being pulled apart. The required contact condition may be imposed to connect between the structure and the rigid support, and specifying a incorporating large, and zero local stiffness in compression and tension respectively.

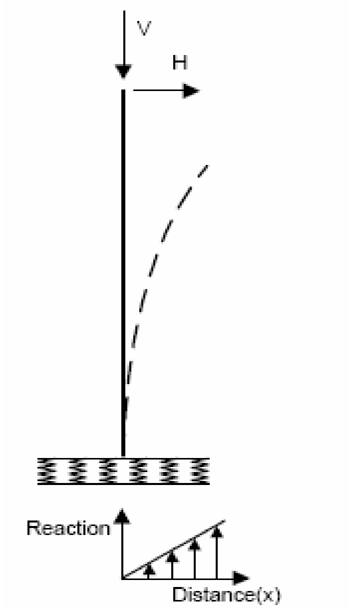


Figure 2.3 Example of nonlinear boundary condition

2.5 Materially Nonlinearity

Materially nonlinear effects occur from a nonlinear constitutive model which has disproportionate stresses and strains. Common examples of nonlinear material behaviour are the plastic yielding of metals, the ductile fracture of granular composites such as concrete or time-dependent behaviour such as creep.

2.6 Element Library

The main function of element library included in LUSAS is to help to reduce user's time in analysing process. The element library enables coarse meshes to be used in modeling and this will save the user's time in data preparation and also interpretation of result. Even with the coarse meshes, LUSAS can provide a good result.

All elements in LUSAS are included with a shape function. This will reduce the coding effort and also lead to high performance elements. Besides that, all elements in LUSAS have to pass several tests before being accepted. These tests include the patch test for convergence, the patch test for mechanisms, convergence rate tests and comparisons of results with extensive experimental and theoretical results.

The LUSAS Element Library contains more than 100 element types. The elements are classified into groups according to their function. The groups are:

- Bars
- Beams
- 2D Continuum elements
- 3D Continuum elements
- Plates
- Shells
- Membranes
- Joints

Followings are some brief explanation of the element groups:

(a) Bar Elements

Bar elements are used to model plane and space truss structures, and stiffening reinforcement. There is 2D and 3D bar elements in LUSAS which can either be straight or curved. Bar elements model axial force only.

(b) Beam Elements

Beam elements are used to model plane and space frame structures. LUSAS beam elements may be either straight or curved. Beam element can model axial force, bending and torsion behavior.

(b) 2D Continuum Elements

2D continuum elements are used to model solid structures that can be assumed to be 2-dimensional. 2D continuum elements may be applied to plane stress, plane strain and axis-symmetric solid problems.

(d) 3D Continuum Elements

3D continuum elements are used to model fully 3-dimensional structures.

(e) Plate Elements

Plate elements are used to model flat structures which deformation can be assumed as flexural. Both thin and thick plate elements are included in LUSAS. Triangular, quadrilateral and ribbed flexural plate elements are also available.

(f) Shell Elements

Shell elements are used to model 3-dimensional structures which behaviour is depend on both flexural and membrane effects. Both flat and curved shell elements, either triangular or quadrilateral, thin or thick elements are available in LUSAS.

(g) Membrane Elements

Membrane elements are used to model 2 and 3-dimensional structures which behaviour is depend on in-plane membrane effects. LUSAS includes both axis-symmetric and space (3-dimensional) membrane elements.

(h) Joint Elements

Joint elements are used to model joints between LUSAS elements. Joint elements may also be used to model point masses, elastic-plastic hinges, or smooth and frictional element contacts.

(i) Field Elements

Field elements are used to model quasi-harmonic equation problems such as thermal conduction or potential distribution. LUSAS includes bar, plane, axis- symmetric solid and 3-dimensional solid field elements.

(j) Interface Elements

These elements should be used at places of potential delamination between 2D continuum elements for modeling delamination and crack propagation.

Table 2.1 Element groups in LUSAS

Element Group	Element Subgroup	LUSAS Element Name
Bars	Structural bars	BAR2, BAR3, BRS2, BRS3
Beams	Engineering beams	BEAM, BMS3, BTS3, GRIL, BRP2
	Kirchhoff beams	BM3, BMX3, BS3, BS4, BSX4
	Semiloof beams	BSL3, BSL4, BXL4
2D Continuum	Plane stress continuum	TPM3, TPM6, QPM4, QPM8, QPM4M, TPK6, QPK8, TPM3E, QPM4E
	Plane strain continuum	TPN3, TPN6, QPN4, QPN8, QPN4M, TNK6, QNK8, TPN3E, QPN4E
	Axisymmetric solid continuum	TAX3, TAX6, QAX4, QAX8, QAX4M, TXK6, QXK8, TAX3E, QAX4E
	Fourier ring	TAX3F, TAX6F, QAX4F, QAX8F
3D Continuum	Solid continuum	TH4, TH10, PN6, PN12, PN15, HX8, HX16, HX20, HX8M, PN12C, HX16C, TH4E, PN6E, HX8E
Plates	Isoflex plates	TF3, QF4,
	Mindlin plates	TTF6, QTF8
	Ribbed plates	TRP3, RPI4
Shells	Axisymmetric shells	BXS3
	Flat thin shells	TS3, QSI4
	Flat box shells	SHI4
	Semiloof shells	TSL6, QSL8
	Thick shells	TTS3, TTS6, QTS4, QTS8
Membranes	Axisymmetric membranes	BXM2, BXM3
	Space membranes	TSM3, SMI4
Joints	2D joints	JNT3, JPH3, JF3, JRP3, JAX3, JXS3
	3D joints	JNT4, JL43, JSH4, JL46, JSL4
Field	Thermal bars	BFD2, BFD3, BFX2, BFX3, BFS2, BFS3
	Thermal links	LFD2, LFX2, LFS2
	Plane field	TFD3, TFD6, QFD4, QFD8
	Axisymmetric field	TXF3, TXF6, QXF4, QXF8
	Solid field	TF4, TF10, PF6, PF12, PF15, HF8, HF16, HF20
Interface	2D Interface	IPN6
	3D Interface	IS16

2.7 Plane Stress

Plane stress is a situation where a thin plate is loaded by forces applied at the boundary, parallel to the plane of the plate and distributed uniformly over the thickness, the stress components are zero on both faces of the plate. It can be assumed that these three components are independent of z as they do not vary through the thickness (Timoshenko and Goodier,1970).

Plane stress typically occurs in thin flat plates that are acted upon only by load forces that are parallel to them. In certain situations, a gently curved thin plate may also be assumed to have plane stress for the purpose of stress analysis. This is the case, for example, of a thin-walled cylinder filled with a fluid under pressure. In such cases, stress components perpendicular to the plate are negligible compared to those parallel to it (Meyers and Chawla,1983).

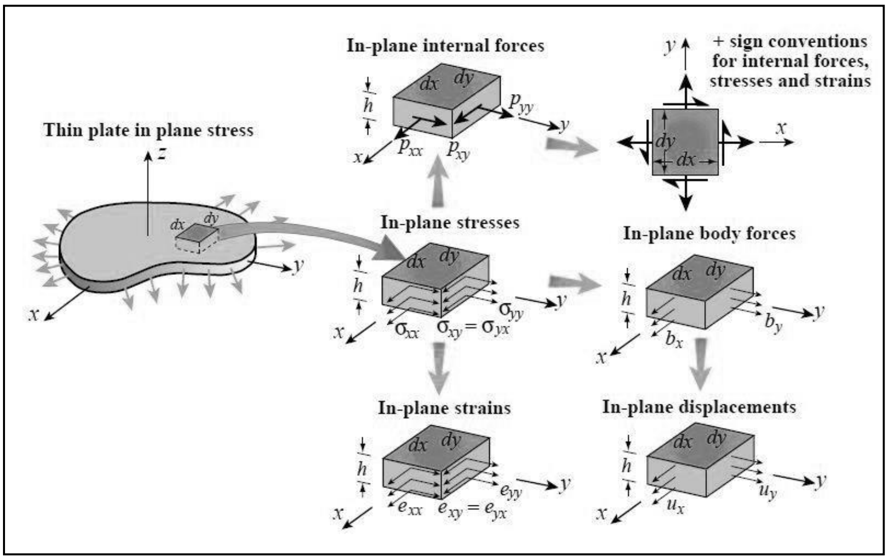


Figure 2.4 Notational conventions for in-plane stresses, strains, displacements and internal forces of a thin plate in plane stress

This state of strain is applicable for long prismatic structures restrained at the ends and subjected to loads which act normal to the longitudinal axis and are invariant with respect to it. Practical examples include a long pipe carrying pressurized fluid, a roller bearing, and the culvert of a dam (with longitudinal ends usually bounded by hillocks) (Bashkar and Varadan, 2009).

2.8 Plane Strain

The strain compatibility equation

$$\varepsilon_{x,yy} + \varepsilon_{y,xx} = \Upsilon_{xy,xy}$$

along with the strain-stress relations

$$\varepsilon_x = (\sigma_x - \mu\sigma_y) / E \; ; \; \varepsilon_y = (\sigma_y - \mu\sigma_x) / E \; ; \; \tau xy / G = 2(1 + \mu)\tau xy /E.$$

The equilibrium equations are needed together with the stress approach and it requires the solution of the above compatibility equation. It is still possible, using the equilibrium equations, to rewrite the above compatibility equation in a simpler form. One has to differentiate the first equilibrium equation with respect to x, and second with respect to y, and sum of the two.

A plane strain formulation of the rate constitutive law for standard elastic-plastic materials was proposed by Corradi and Gioda(1979). Function of the remaining stress and of the non-vanishing plastic strain components been defined as the plane strain hypotheses and the elastic portion of the constitutive relation permitted the expression. In the same manner, in-plane yield function was defined on this basis, which did not depend explicitly on stress, and it was shown that the associated flow rule remained valid provided the actual plastic strains were replaced by equivalent in-plane measures, counting for plasticity in the transverse direction. The effects of transverse yielding showed as a fictitious hardening in-plane representation.

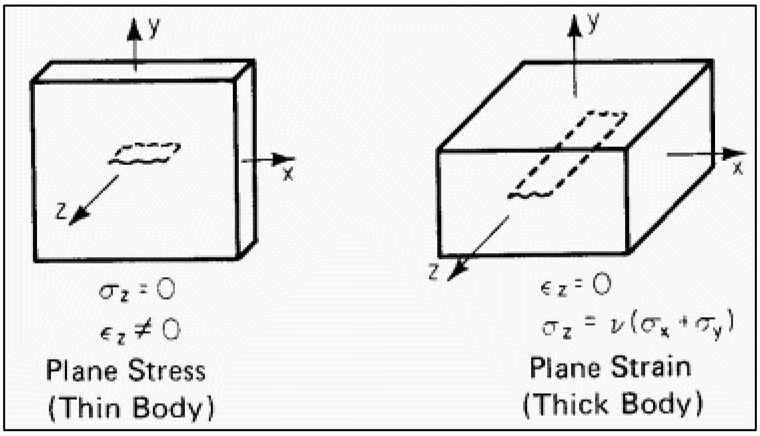


Figure 2.5 Difference in the plane stress and strain body

2.9 Design and Construction of a Structure in Visual Concrete

2.9.1 Concrete Characteristic

It can be said that the most important part of a construction and structure is the concrete or in visual, the visual concrete. Visual Concrete must be measured correctly and measured up to demanding aesthetic standards (Heinle,2000).

2.9.2 Strength and Plasticity

The crucial benefits for concrete are its great compressive strength, which allow the use of bold, slender components and almost unlimited range of forms which is possible to, both in overall appearance and in detail, its variable surface texture; and its range of colours (Heinle,2000).

2.9.3 Ductility

Ductility is the capacities of a material undergo large inelastic deformations without failure. The opposite for a ductile material is brittle material. (Meyer,2013).

Concrete can also resist to weather and frost. It can also resist to the polluted weather if it is made very well and denser enough with resistant type of cement. Concrete offers good protection against fire, provided that the proportion of reinforcement forms less than 2% of the cross-section. Its toughness and resistance to abrasion make for durability. Even if pieces of a concrete are knocked away, the effect of so homogenous a material is less disturbing than it is on (Heinle,2000).

2.10 Plasticity

2.10.1 The von Mises Yield Criterion

The von Mises stress is often used in determining whether an isotropic and ductile metal will yield when subjected to a complex loading condition. This is accomplished by calculating the von Mises stress and comparing it to the material's yield stress, which constitutes the von Mises Yield Criterion. Von Mises stress is considered to be a safe haven for the engineers. A design can be considered fail when the maximum value of Von Mises stress induced in the material is more than strength of the material. Von mises can works very well in most of the cases and can work better when a material is already ductile in nature.

The objective is to develop a yield criterion for ductile metals that works for any complex 3-D loading condition, regardless of the mix of normal and shear stresses. The von Mises stress does this by boiling the complex stress state down into a single scalar number that is compared to a metal's yield strength, also a single scalar numerical value determined from a uniaxial tension test on the material in a lab.

Concept of Von Mises stress comes from the distortion energy failure theory where when the distortion energy in the real case is more than any simple tension case at the period of failure. Distortion energy required per unit volume, u_d for a general 3 dimension case is given in terms of principal stress values as

$$u_d = \frac{1+\nu}{3E}\left[\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}\right] \qquad \textbf{(2)}$$

The condition of failure will be as follows,

$$\left[\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}\right]^{1/2} \geq \sigma_y \qquad (3)$$

Left hand side of above equation is denoted as Von Mises stress,

$$\left[\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}\right]^{1/2} = \sigma_v \qquad (4)$$

Thus the failure can be simplified as $\sigma_v \geq \sigma_y$ ıere it will be the criterion for a design engineer to check.

Von Mises is an adequate scalar measure of complex stress fields for ductile materials. Von Mises stress is the accepted measure to determine the onset of plastic behavior of metallic materials, which is an important issue in the design of both civil and aerospace structures. Multiaxial fatigue is considered governed by the equivalent Von Mises stress.

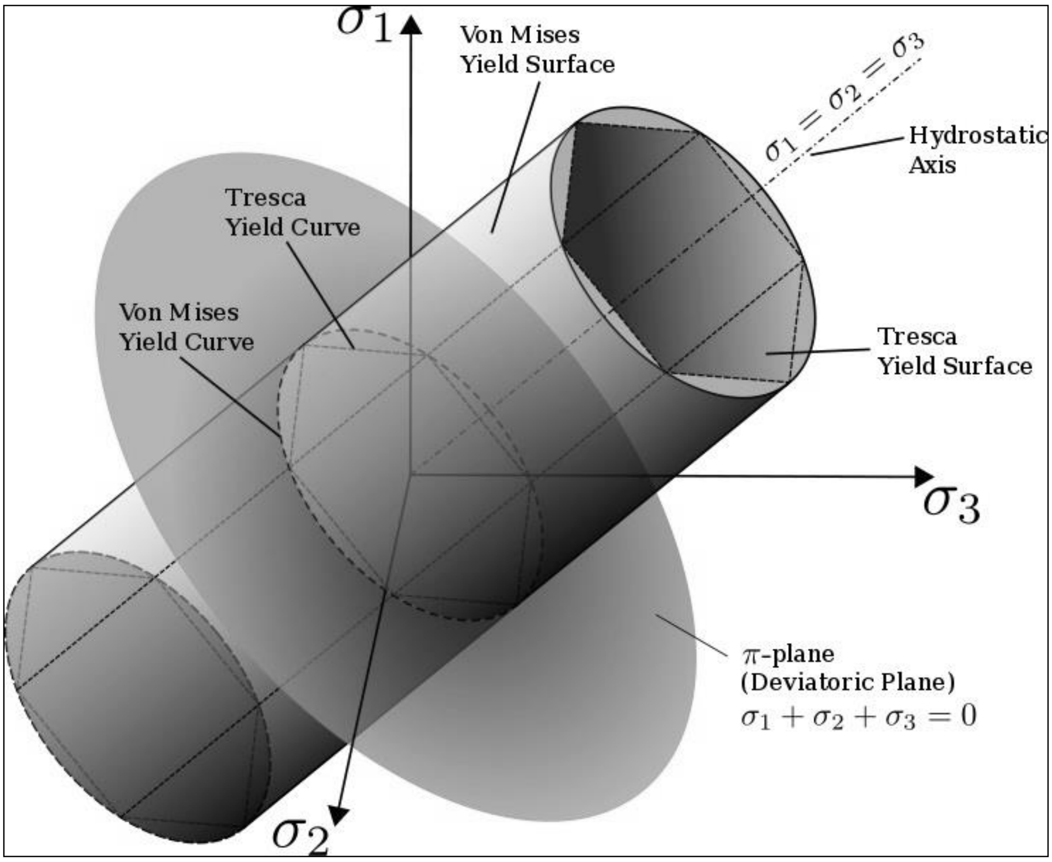


Figure 2.6 The von Mises yield surfaces in principal stress coordinates.

2.10.2 Flow Rules

When subjected to an external loading, the geometry of a material will change and this is termed as the deformation. It can be an elastic or plastic deformation depending on the material behaviour after the removal of the loading. Elastic is the deformation that is reversible while plastic is irreversible deformation (Henry, 2009).

Stress-strain relationship is the major concern when studying on material behaviour. Constitutive equations are known as the equations that describe these relationships.

It consists of:

i. The stress state when yielding occurs is determined by the yielding criterion.
ii. Flow rule is used to describe the increment of plastic strain when yielding occurs
iii. A strain hardening rule to describe how the material is strain hardened as the plastic strain increases. (Henry,2009)

2.10.3 Idealization of Stress Strain Curve

It has become customary to idealize the curve in various way because of the complex nature of the stress strain curve.

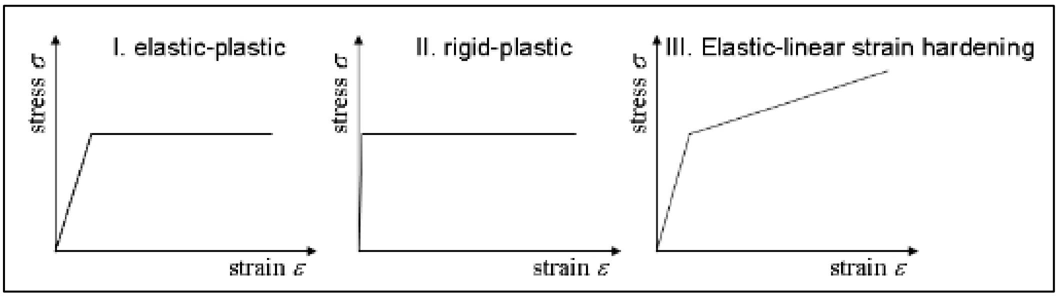


Figure 2.7 Idealized stress-strain curve

A perfectly plastic material shows an unlimited amount of deformation or strain, at constant stress and after the removal of load, the plastic strain or deformation cannot be recovered. The illustration has been illustrated in Figure 2.7 for an ideal elastic-plastic material. The rigid idealization is valid when the deformation is negligibly small and for linear-hardening material, curve III could be a reasonable approximation.

A typical elastic-linear strain hardening curve is shown in Figure 2.8, where the stress in y is the yield stress and E is the Young Modulus, E in terms of t if the tangential modulus after yielding. If a strain hardened material is unloaded from the elastic-plastic range of deformation and reloaded again, the plastic deformation will not resume at the stress level indicated by intital yield point, but at the stress level reached just before the unloading (Henry,2009).

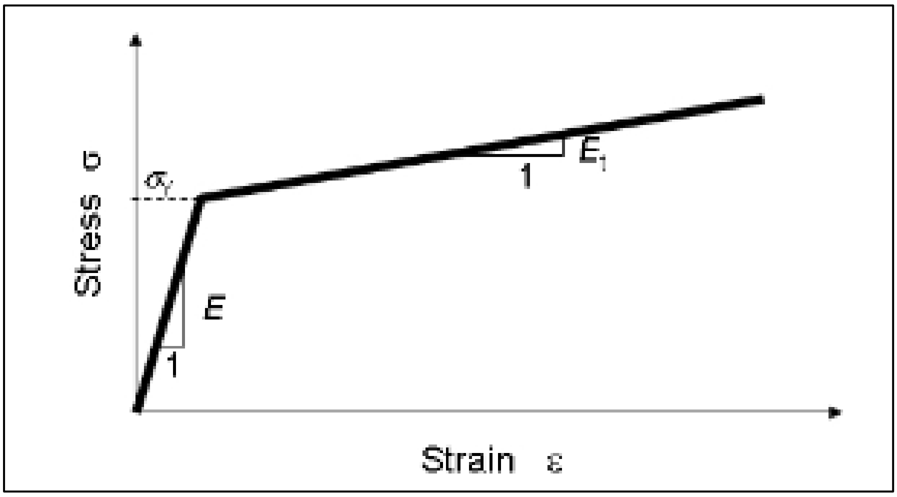


Figure 2.8 Stress- strain curve from a unixial test

2.10.4 The Tangent and Plastic Modulus

Stress and strain are related in the elastic region where the Young Modulus is multiplied to the strain to get the stress. The tangent modulus is the slope of the stress-strain curve in the plastic region and will be the general change during the deformation where $\delta\sigma = K\delta\varepsilon$ is the relationship between the general increment.

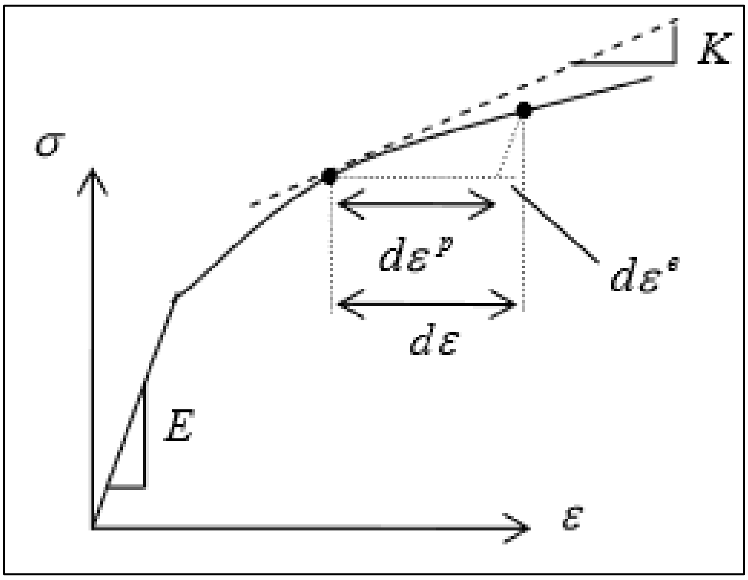


Figure 2.9 The Tangent Modulus

After the yield, the strain increment consists of both elastic, $\varepsilon^{\varepsilon,}$ and plastic, $d\varepsilon^{p}$, strains :

$d\varepsilon = d\varepsilon^{e} + d\varepsilon^{p}$ (5)

The stress and plastic strain increments are related by the plastic modulus H:

$d\sigma = H + d\varepsilon^{p}$ (6)

and it follows that $1/K = 1/E + 1/H$ (Kelly,2008).

2.11 Research Study

2.11.1 Finite Element Analysis on von Mises Stress Distribution of SI DSP

The paper describe a 2D axisymmetric quasi-static finite element model based on 300 mm wafer and double-side polishing (DSP) using a COMSOL Multiphysics software. The observation has been made on the effects of Young's Modulus and Poisson ratio of polishing pad and the thickness ratioof upper and lower pads on the von Mises stress distribution (Wang, 2012).

Based on the paper, it shows that the von Mises distribution on the water surface with vary number of Young Moduli ad Poisson's ratio of 0.1. The amplitude of the stress peak is smaller and also the curve levels down with the increase of Young's Modulus of pad. The results are consistent which shows the deformation of pad elastic was affected by the Young Modulus. Another results show that Poisson's ratio of the pad has a lesser effect on the von Mises stress distribution on the wafer surface. (Wang, 2012).

2.11.2 General Report on Local Ductility

Poor local ductility of steel structures contributed by a series of factor which was observed during recent earthquakes in the USA and Japan. These factors are explained in the thesis which said that the factor on the resistance side are because of the differences in in real and design yield stress. Another contributing factor include the thickness resistance of the steel and the past underestimates the need of plastic rotations (Andre,2000).

The capacity design cannot be based on the nominal strength or a lower characteristic value of the yield strength without forgetting the neighboring connection. Before the modern era, the structural engineers have calculated the proper moment frames by sizing the members for strength and drift using code forces and then increasing the strength of the members. The engineer manage to increase the strength by designing connections corresponding to certain prescribed types (Andre,2000).

CHAPTER 3

METHODOLOGY AND ANALYSIS

3.1 Introduction

Generally this research is made to investigate the strength and stress behaviour using LUSAS Software. However, this research will be limited to several aspects. Thus, methodology is very important in order to run this research in smooth without any major issues in the future days. This will help in getting a very good data so that a detailed discussion will be shown in an interactive ways. Refer Figure 3.1. Proper guidance documentation also required to ensure the development activities is comply with all standard requirement. The methods that applied in this study are based on three (3) stages which are:-

Stage 1: Analysis on reinforced concrete beam using computer software

Stage 2: Collect data

Stage 3: Documentation

3.2 Early Study

In order to get some of the general information about the main topic, a literature review has been done. This study has been progress widely and systematically to ensure that every detail of data was obtained and further study was done. This was known as the earlier study. The clearer plan of research was form regarding the scope of study through the literature review and so, the study was executed in order. Earlier data and general information is available from the previous journal, some from the internet, elaboration, research that has been made, magazines, books and newspapers. Most of the information was obtain from the internet and several articles were got from the personal publications.

These are some of the information that were found during the literature review

I. Reinforced concrete and its characteristic
II. Ultimate limit state of reinforced concrete.
III. The awareness about ductility issue.
IV. Current issues.

3.3 Scope of Study and Data Collection

Data collection is another important factor in doing this study. Since it cannot be written through its theory only, interview, survey and case study should be done based on the objectives and scope of study. Computer program called London University Stress Analysis System or LUSAS will be fully utilized throughout the investigation of this project.

LUSAS was developed by research workers from the University of LONDON. The team was led by Dr. Paul Lyons. LUSAS is a UK-based developer and supplier of Finite Element Analysis(FEA) application software products. LUSAS can manage many types of linear and non-linear stress, dynamics, composite and thermal engineering analysis problem. It can be used on all types of structures from buildings, grandstand and towers through to dams, docks and tunnels. It have the capabilities of linear, nonlinear, staged construction, soil-structure interaction, creep, pre-stress, post tensioning, seismic, blast, buckling, impact, fire and fatigue analysis.

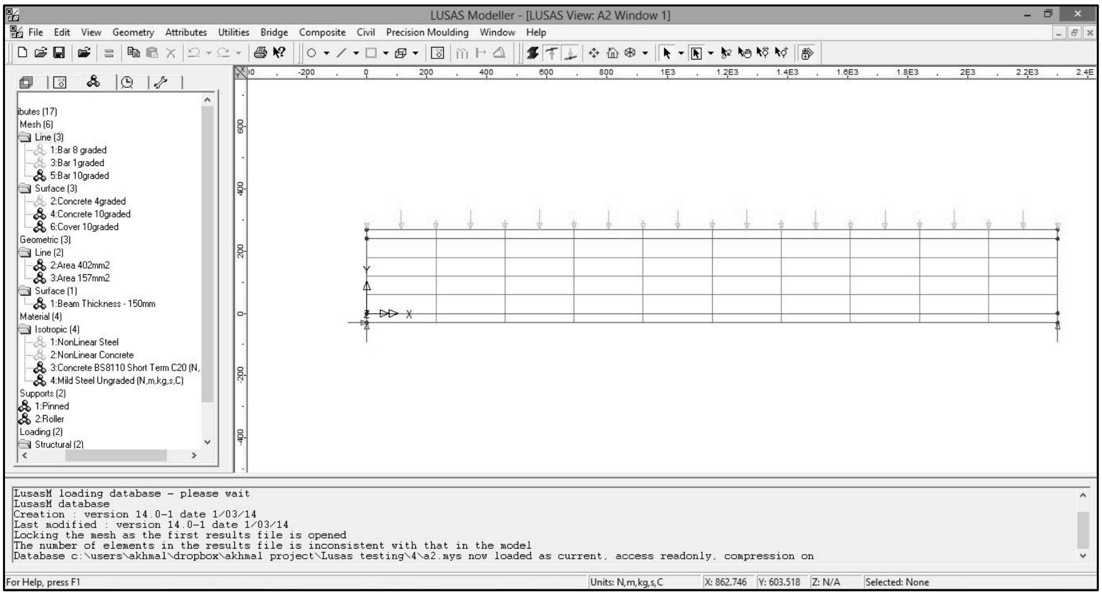


Figure 3.1 User-interface of LUSAS Software

All laboratory data was taken from ‘Shear Behaviour of Reinforced Concrete Beams Strengthened With Carbon Fibre Reinforced Polymer Fabrics’ by Mohd Yunus (2012) as the controlled results.

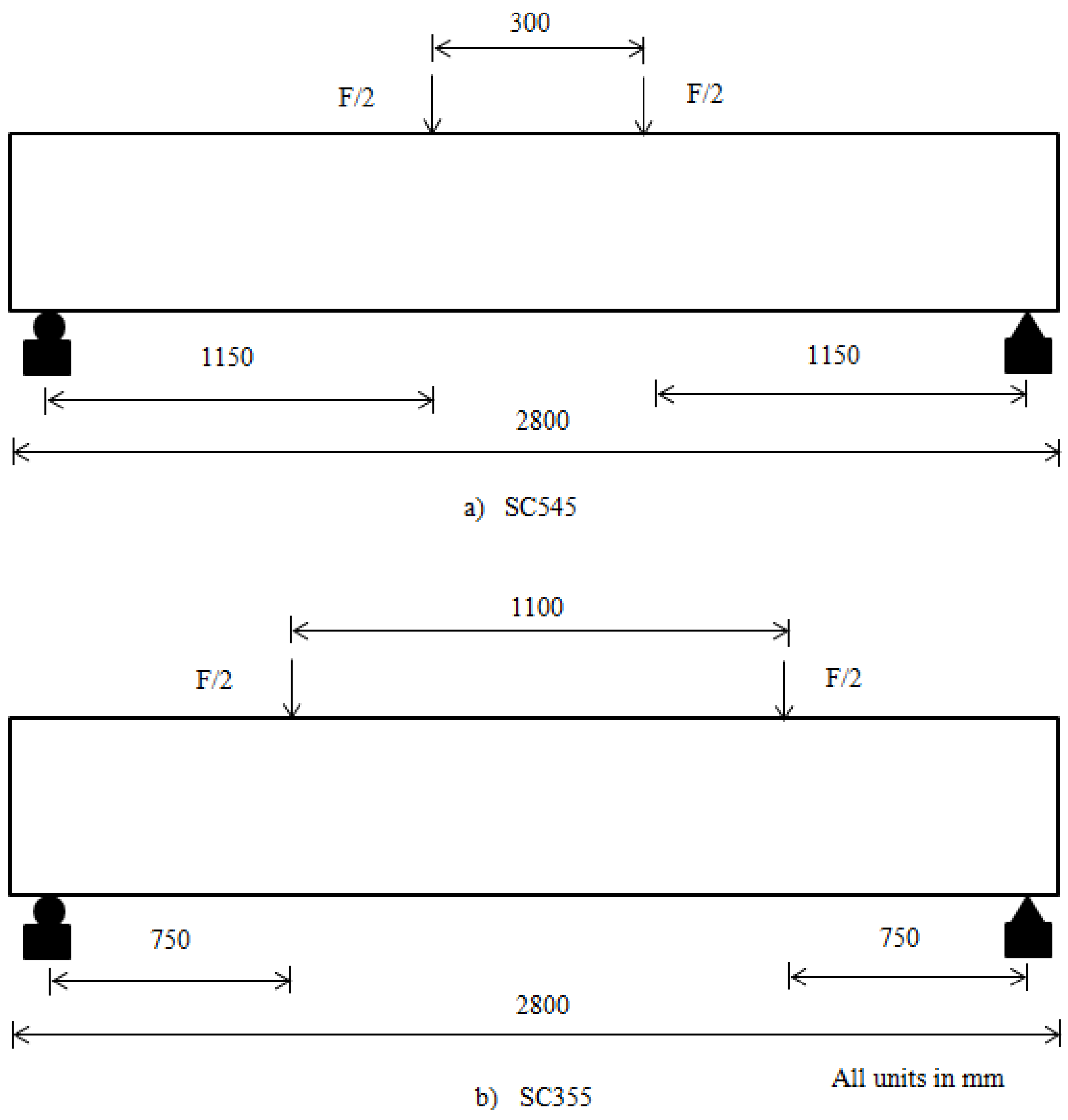


Figure 3.2 Schematic of beam under point load test for beam a) $a_{v/}d$ =5.45 b) a_v/d = 3.55 (Mohd Yunus,2012)

Table 3.1 Attributes used in LUSAS Software

Attributes	**Element**
Line	2 Dimensional Quadratic Bar
Surface	Quadrilateral Quadratic Plane Stress
Geometric – Line	$402mm^2$ (Steel Reinforcement Area)
Geometric – Surface	120mm
Material – Isotropic	Nonlinear Steel (Stress Potential – von Mises) Nonlinear Concrete (Concrete (model 94))
Loading	1kN
Support – End of Beam	Pinned
Support – End of Symmetrical Part	Roller in X Direction

Table 3.2 Mechanical properties of tensile tests

Rebar Diameter	**Yield stress (Mpa)**	**Yield Strain (microstrain)**	**Ultimate Stress**	**Young Modulus (E) (Gpa)**
T16	589	3000	668	206

Table 3.3 Concrete strength of the beam tested

Beam	**Concrete cube strength (Mpa)**	**Modulus of rupture, f_r**	**Young Modulus (E) (Gpa)**
SC545	35.7	3.74	27
SC355	35.7	3.74	27

This beam will be visualized in LUSAS Software and the load deformation graph will be created to be compared with the controlled results. Stress contour will be plotted from LUSAS in order to analyze the plastic behavior and failure type of the concrete beam model to verify its result with experimental data.

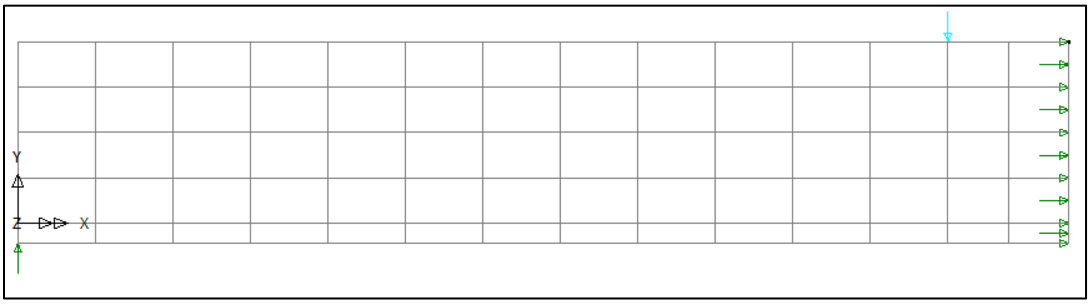


Figure 3.3 Model of concrete beam after all attributes has been assigned

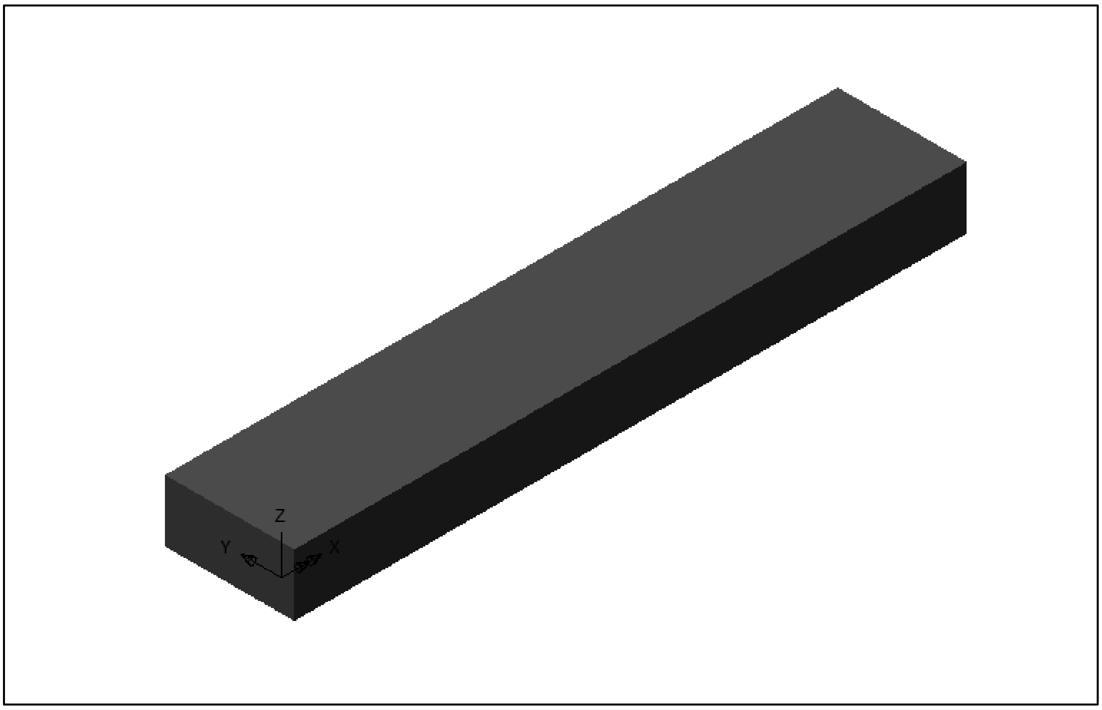


Figure 3.4 Visualisation of reinforced concrete beam model in LUSAS Software

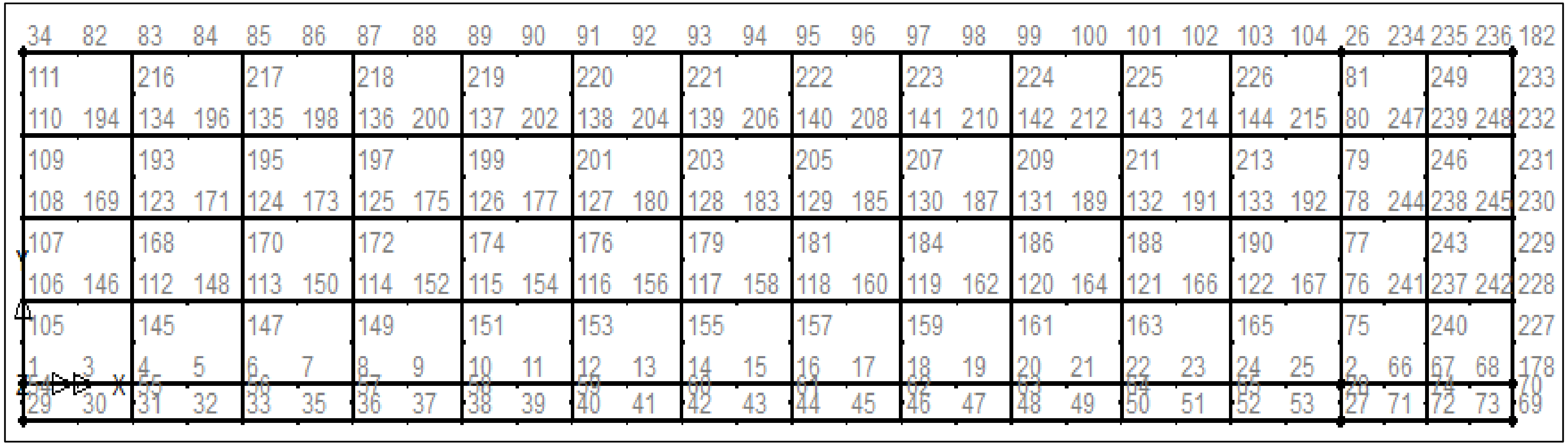


Figure 3.5 Node numbering for beam model specimen SC545

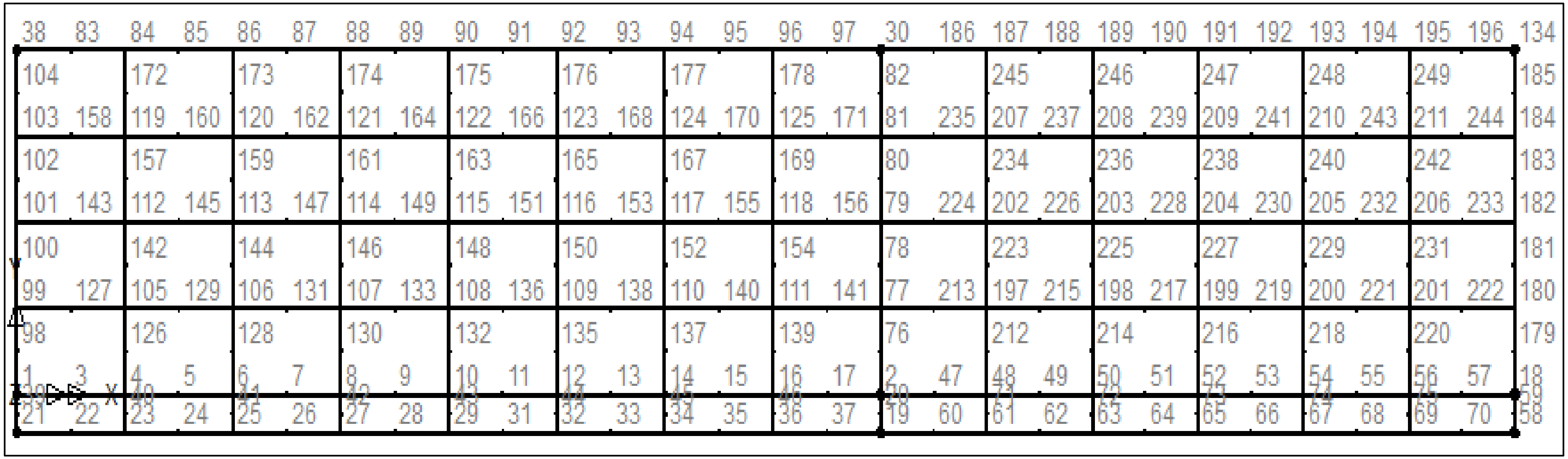


Figure 3.6 Node numbering for beam model specimen SC355

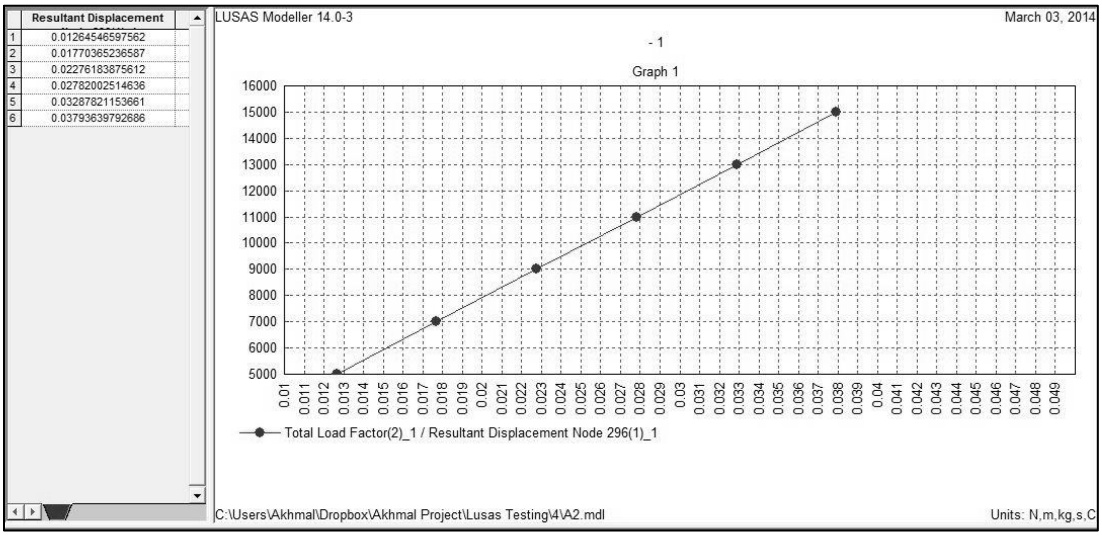


Figure 3.7 Example of Load- Displacement Graph using LUSAS Software

3.3.1 Case Study

The case study regarding the management of reinforced concrete and its behaviors are the data collected in this study. The information needed from this study is the laboratory value of reinforced concrete testing and other relevant information.

3.4 Data Analysis

Analyzing data is conducted to determine the results from case study activities. Data interpretation was made from the end results. These data was obtains through the analysis, interviews, and literature review of journal paper, company report and etc. At this stage, all the data will be evaluate and choose. Any data that not comply with the study objective will be eliminated. The appropriate data were analyses and the results were shown in tabulate form, graphs and figures. The end results from the analysis were documented as a guidance document to achieve the objective of the study. The documentation consist the literature review, founder, conclusion and the suggestion towards a better building structure.

3.5 Basic Concept of LUSAS Software

Before proceed to the analysis of the structure, it has to be model first. All the data needed to be entering into the LUSAS Software in order for the software to be run perfectly and give the precise result. These are the criteria that need to be done:

a) Meshing

Meshing is a stage where the entire model in terms of geometric features which must be sub-divided into finite elements for solution. Every lines, surfaces or volumes have different type of meshing. For meshing a volume, the element selected must be defined in terms of its generic element type, element shape and interpolation order. In this project, both bar graded and and plane stress element is mesh differently.

Table 3.4 Meshing for modeled structure

Mesh	**Element type**	**Dimension**	**Interpolation order**
Line	Bar	2 Dimensional	Quadratic
Surface	Plane Stress	Quadrilateral	Quadratic

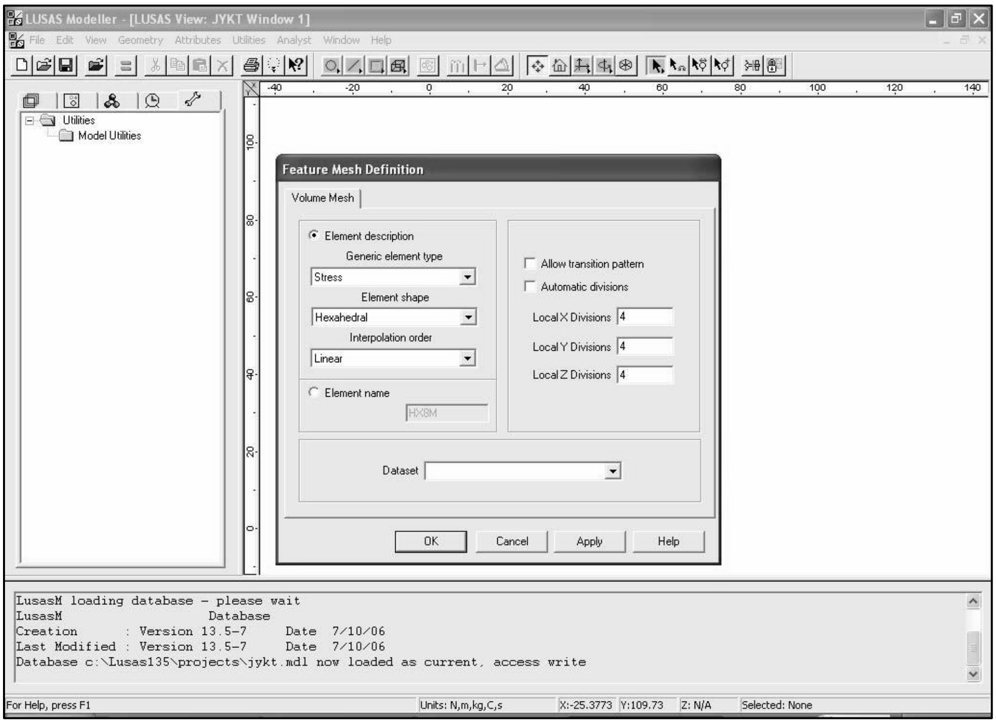


Figure 3.8 LUSAS Interface for Meshing

b) Geometry

This stage is to give geometric attributes such as thickness, cross sectional area, second moment of area, torsion constant and others. Most of these properties is taken from the experiment done by Mohd Yunus (2012).

Table 3.5 Values for Geometric in modeled structure

Geometric	**Cross-sectional Area(mm^2)**	**Thickness (mm)**
Line	402	-
Bar	-	120

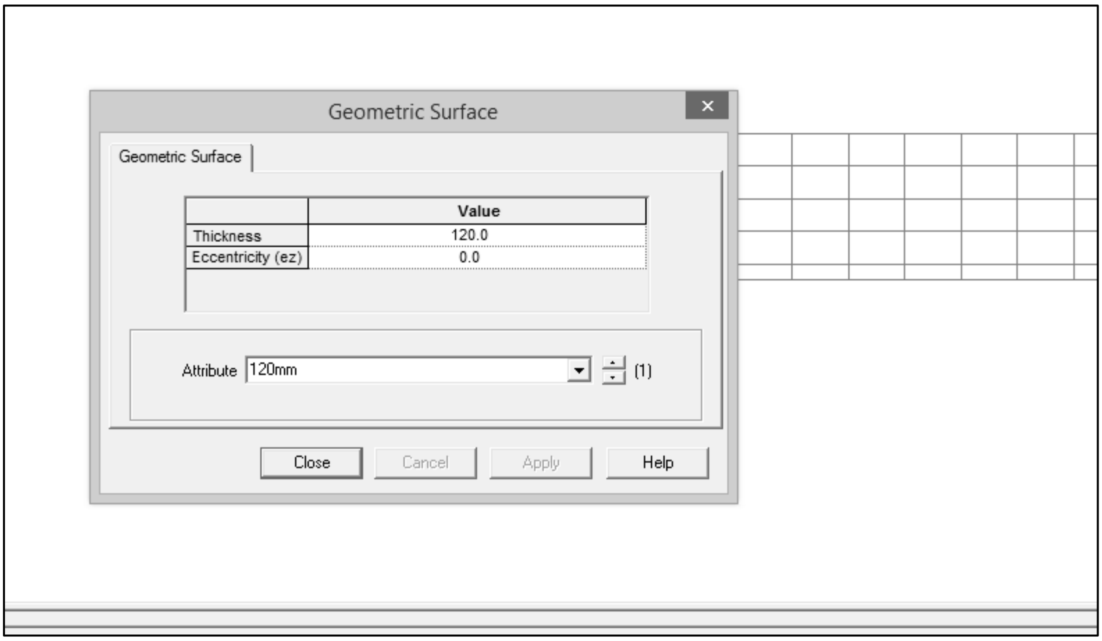


Figure 3.9 Interfaces for Surface Geometric

c) Materials

Materials assign in this model are the best fit according to the experimental materials. All the value for each material taken in the experiment data has been put into LUSAS Software so that we can see the differences in between computing software and experimental data. All the materials are set isotropic for both nonlinear steel and nonlinear concrete.

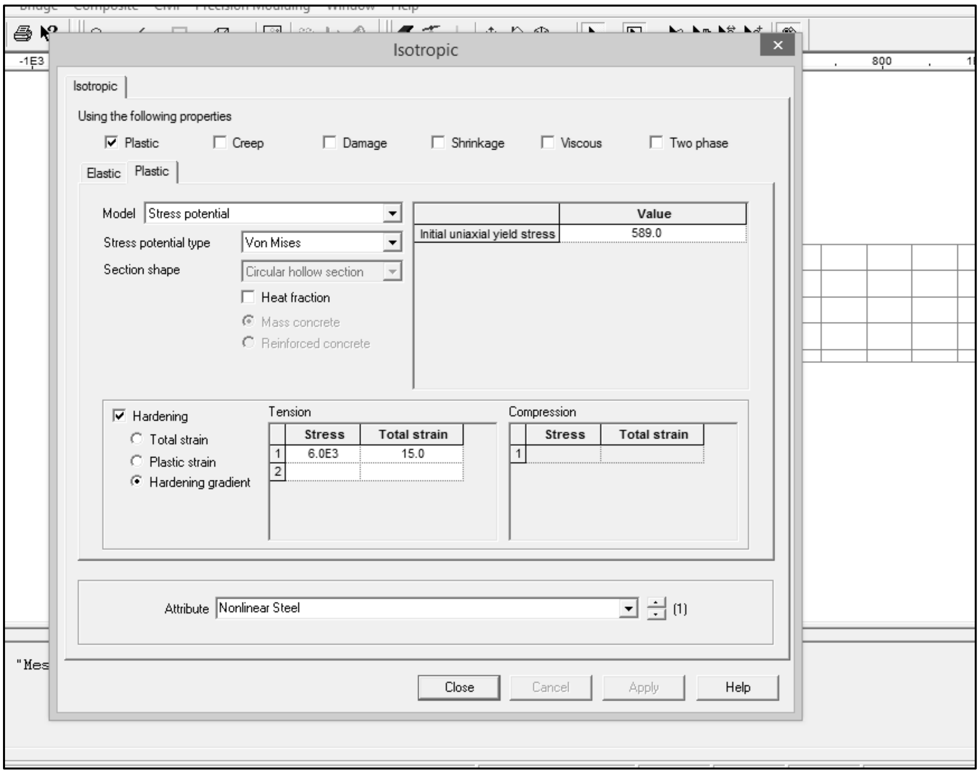


Figure 3.10 Interface for Nonlinear steel

For nonlinear steel material, the analysis is done by using stress potential model and stress potential type will be using von Mises. Von Mises is a part of a plasticity theory that is best to use with ductile materials. All material response will be assumed as elastic prior to yield. The uniaxial yield stress for this material will be 589MPa.

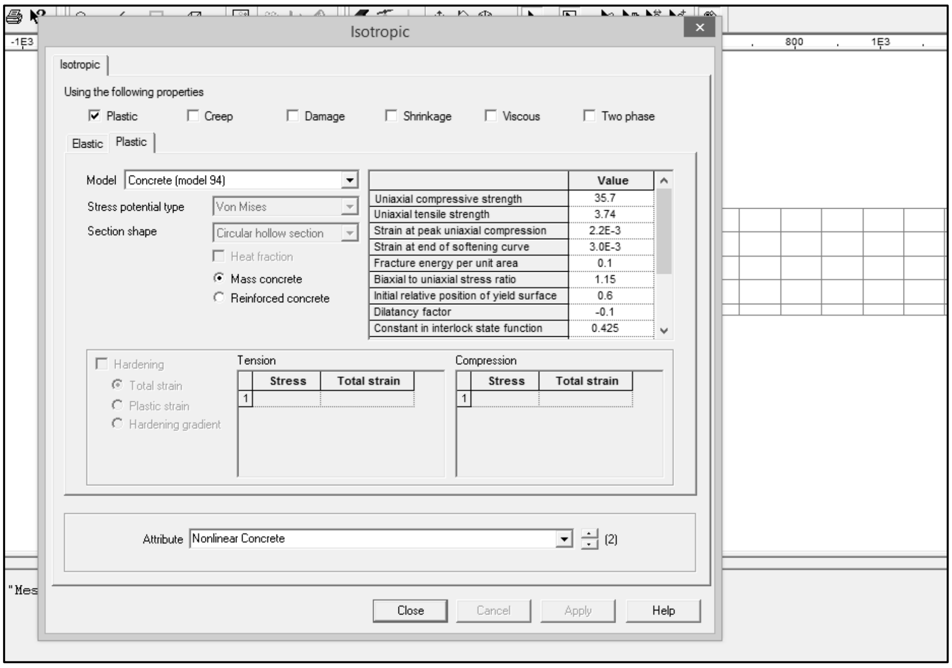


Figure 3.11 Interfaces for Nonlinear Concrete

For nonlinear concrete, structure will be model using concrete (model 94). The uniaxial compressive strength is 35.7 MPa and uniaxial tensile strength are 3.74 MPa

d) Loading

The model is assigned for point load with the value of -1kN as starting load. The value is then increase with increment of 1kN per load factor.

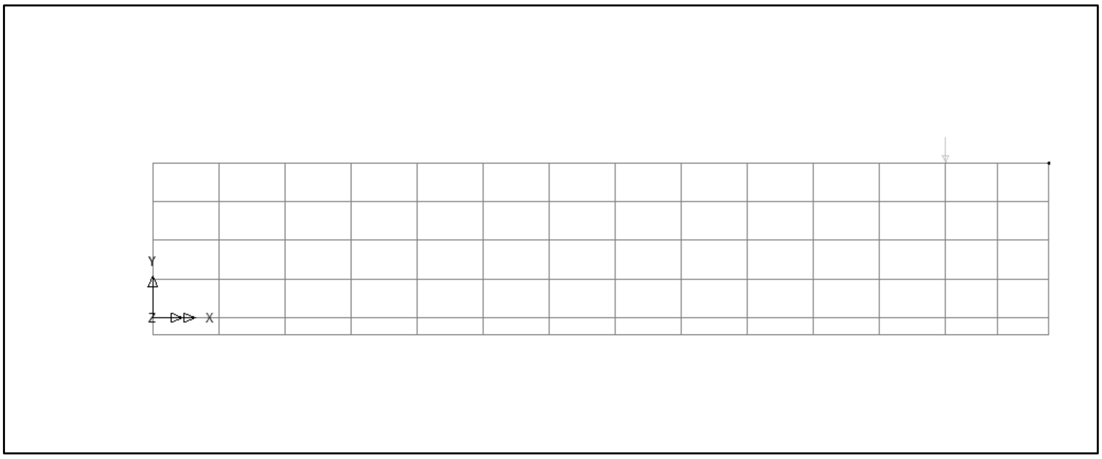


Figure 3.12 Interface for point load of the model

3.6 Conclusion

In this chapter, the methodology to conduct this study and to retrieve information has been identified. The theoretical methodology framework is use to determine the important variables that were used in the analysis. Data collection is divided into two methods which are primary and secondary method. Data collection can be done through several ways such as experimental data and from computing software.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Introduction

In this chapter, the result of the model structure is discussed. The entire procedures were done in accordance with the method described in Chapter 3. The results and discussion were discussed according to the objective of the research discussed in Chapter 1 earlier. All the results were thoroughly discussed and conclusions for the research were drawn out later.

Those results includes the graph of load-displacement for both of the beams, strains for steel and concrete for both of the beams, and the stress contour for both of the beams. Based on the analysis too, the stress behaviour for both of the beams were determined using the stress contour. The discussions were drawn out later based on the results from the analysis.

All the result and discussion were made based on the objectives discussed in this research. The model has successfully model in LUSAS Software and nonlinear finite element analysis has been tested. The results are then tested with theoretical and experimental data from Mohd Yunus (2012).

4.2 Non-linear Analysis of Model

Figure 4.1 shows the deformation of the model after load is applied. It shows that the deflection happen at the center of the beam. All the deformation tested only until it reaches its first crack value. This will make it easier for the test to be done while more parameter can be test on the first crack load.

Table 4.1 Comparison on first crack load and deflection of all beams tested.

Beam	First Crack Load (kN) Theory	First Crack Load (kN) Experiment	First Crack Load (kN) LUSAS	Deflection at first crack load (mm)		
				Theory	Experiment	LUSAS
SC545	14.74	8	13.462	1.11	2.96	3.06
SC355	22.61	16	20.462	1.33	3.39	3.88

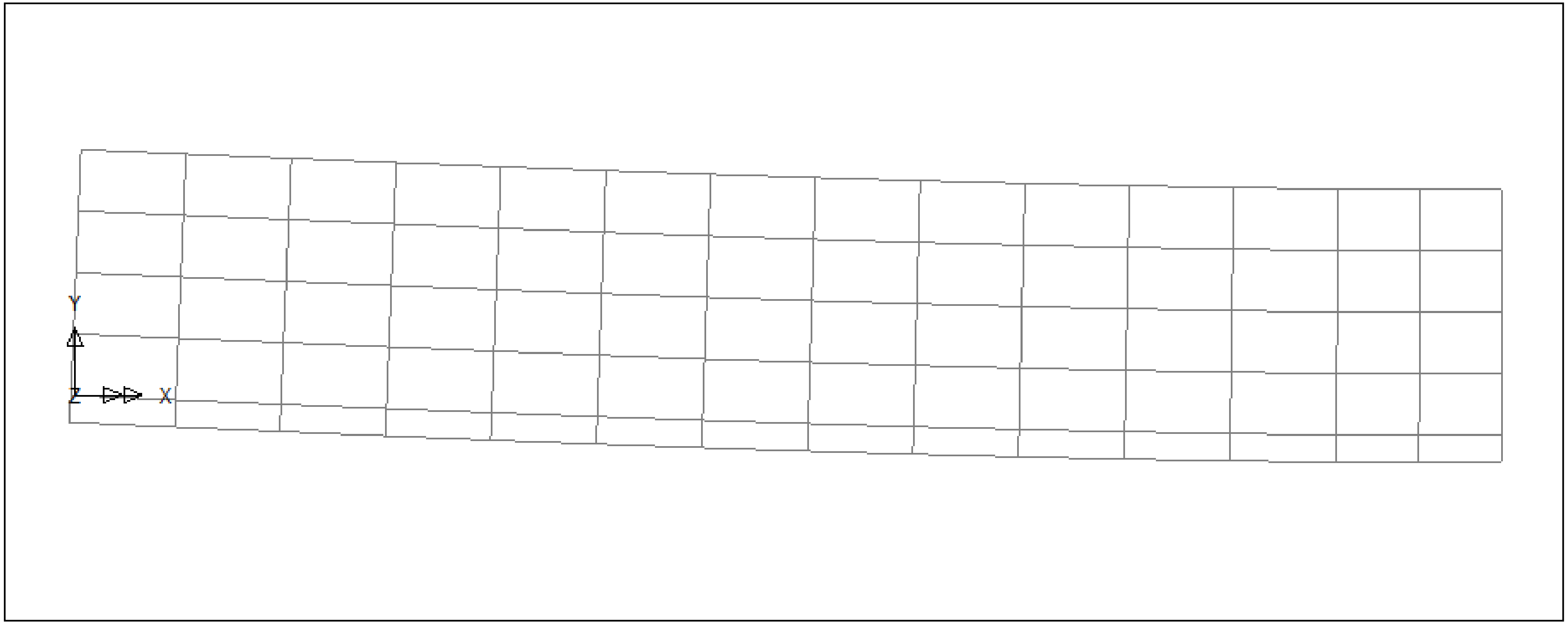


Figure 4.1 Deformation of the concrete beam model

From the Table 4.1, we can conclude that the value for first crack load is quite similar from theory and LUSAS Software. Although the values have quite a difference from the experimental data, Mohd Yunus (2012) said that experiment data could have some error like parallax and etc. This could lead to some error in data taken since the tester of the beam need to be press the load and look at the first crack at the same time. However, the deflection is difference in deflection value is quite big. The theoretical value and experimental value is already having a big difference while LUSAS results have a larger value than both of the results.

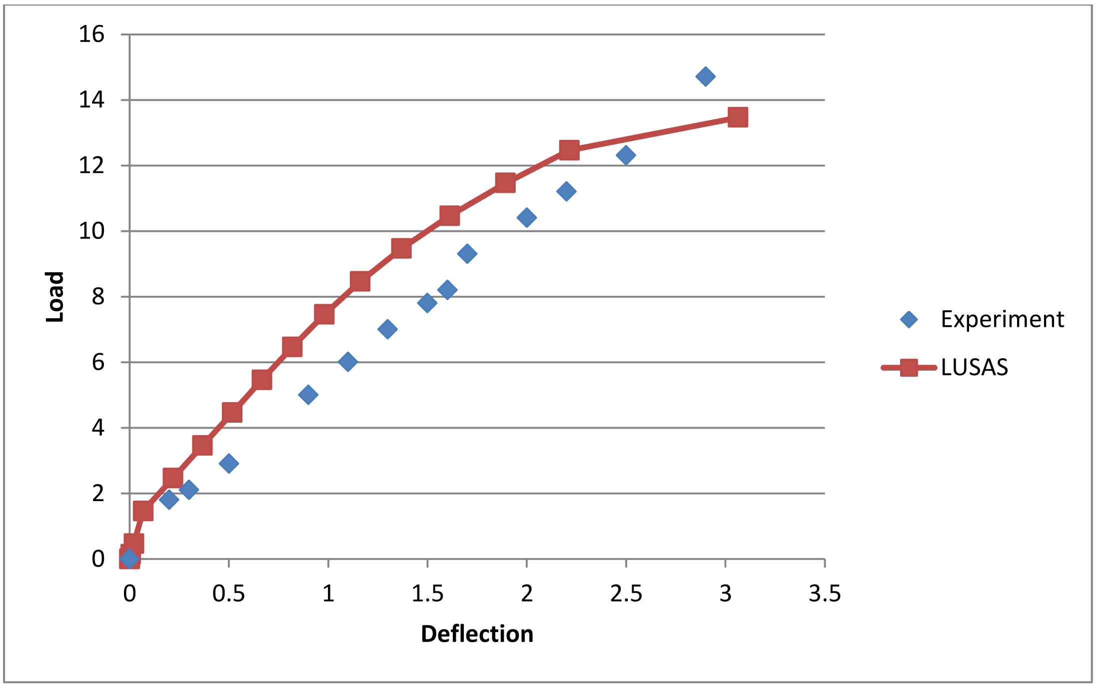


Figure 4.2 Load vs. Deflection for beam specimen SC545

Figure 4.2 shows the differences between LUSAS and experiment data for beam SC545. We can clearly see that the value for the first crack for both type are quite similar which for experiment, first crack happen at 14.7 kN with displacement of 2.96 mm and for LUSAS, first crack happen at the load of 13.24 kN with displacement of 3.06 mm.

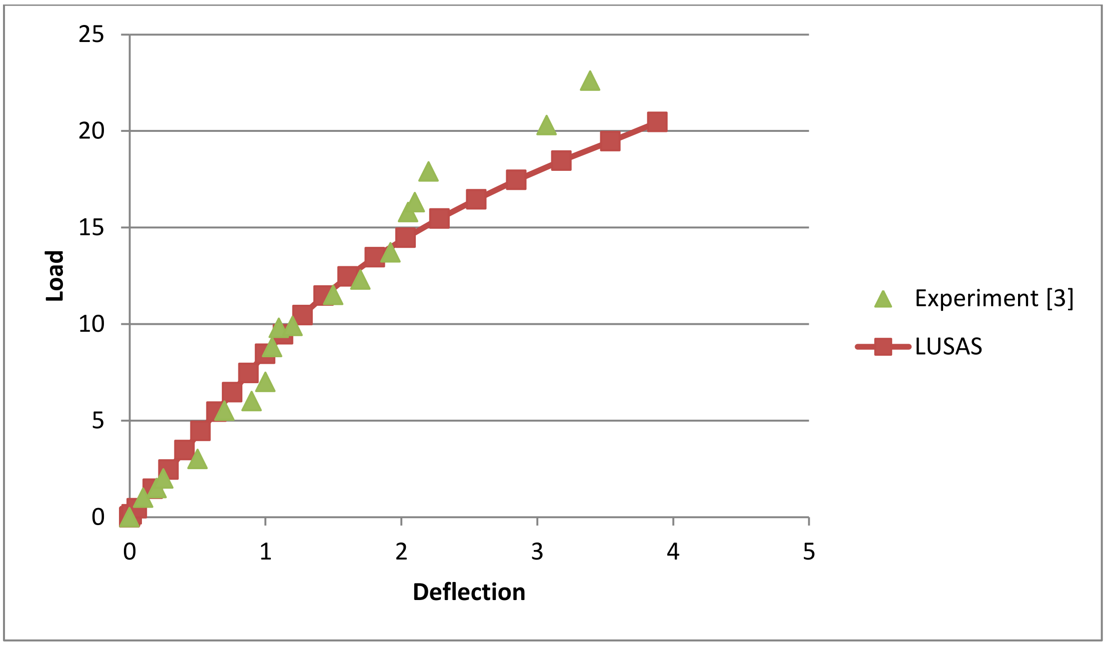


Figure 4.3 Load vs. Deflection for beam specimen SC355

Figure 4.3 shows the differences between LUSAS and experiment data for beam SC355. We can clearly see that the value for the first crack for both type are quite similar which for experiment, first crack happen at 22.6 kN with displacement of 3.39 mm and for LUSAS, first crack happen at the load of 20.46 kN with displacement of 3.88 mm.

Table 4.2 Comparison on Strain value for all beams tested

Beam	Maximum Shear Force Value (kN)	Concrete Strain at First Crack (LUSAS) (10^-3)	Concrete Strain at First Crack (Experiment) (10^-3)
SC545	73.20	0.118	0.19
SC355	73.52	0.117	0.17

Beam	Maximum Reaction (kN)	Steel Strain at First Crack (LUSAS) (10^-3)	Steel Strain at First Crack (Experiment) (10^-3)
SC545	61.26	0.6668	0.85
SC355	61.05	0.6933	0.41

Table 4.2 shows the differences between strain at experiment and LUSAS value at the first crack. For beam SC545, concrete strain value for LUSAS is 0.00012 while for experiment the value is 0.0019. The ratio for LUSAS to experiment value is 0.632. Then, the steel strain value for LUSAS is 0.00067 while for experiment the value is 0.00085. The ratio for LUSAS to experiment value is 0.788. For beam SC355, concrete strain value for LUSAS is 0.00012 while for experiment the value is 0.0017. The ratio for LUSAS to experiment value is 0.706. Then, the steel strain value for LUSAS is 0.00069 while for experiment the value is 0.00041. The ratio for LUSAS to experiment value is 1.68. From the table, we can also get the value for maximum shear force for the beams which is 73 kN and maximum reaction for both beams is 61 kN.

4.3 Behaviour of Concrete Structure Model

The behaviour of concrete beam model is determined using the stress of plane stress at the component of SX. From the plane stress and maximum stress pattern we can prove the type of failure that happened during the experimental test done by Mohd Yunus Ishak (2012). Based on the thesis (2012), beam structure for specimen SC545 failed due to flexural failure with concrete crushing and beam structure for specimen SC355 failed due to shear failure. Both of these specimens have no shear reinforcement and this lead to a lower ductility in the load-displacement graph.

4.3.1 Beam Model for Specimen SC545

From Figure 3.5 from Chapter 3 shows the node numbering for beam model for specimen SC545. The point load is applied at the node number 26, and the roller support is placed at node number 29. Node number 69 shows the bottom center part of the beam or at the end of this model.

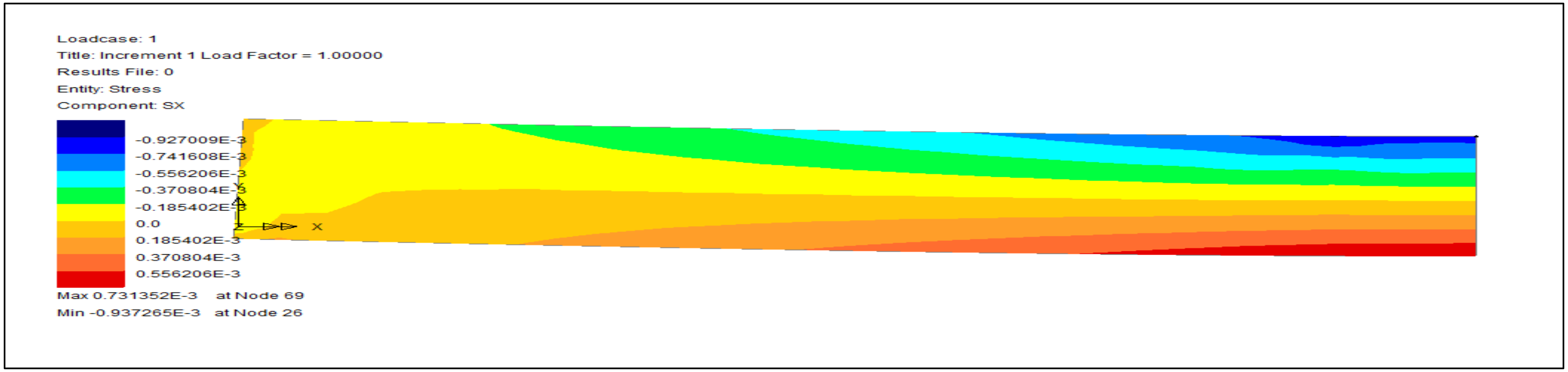


Figure 4.4 Stress SX Contour for load 1N (Beam SC545)

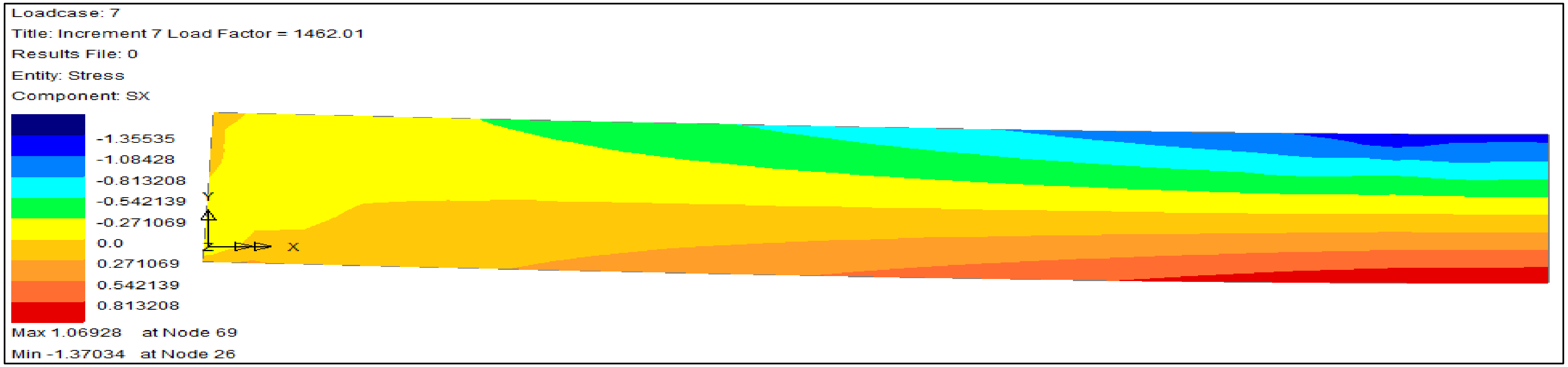


Figure 4.5 Stress SX Contour for load 1.46kN (Beam SC545)

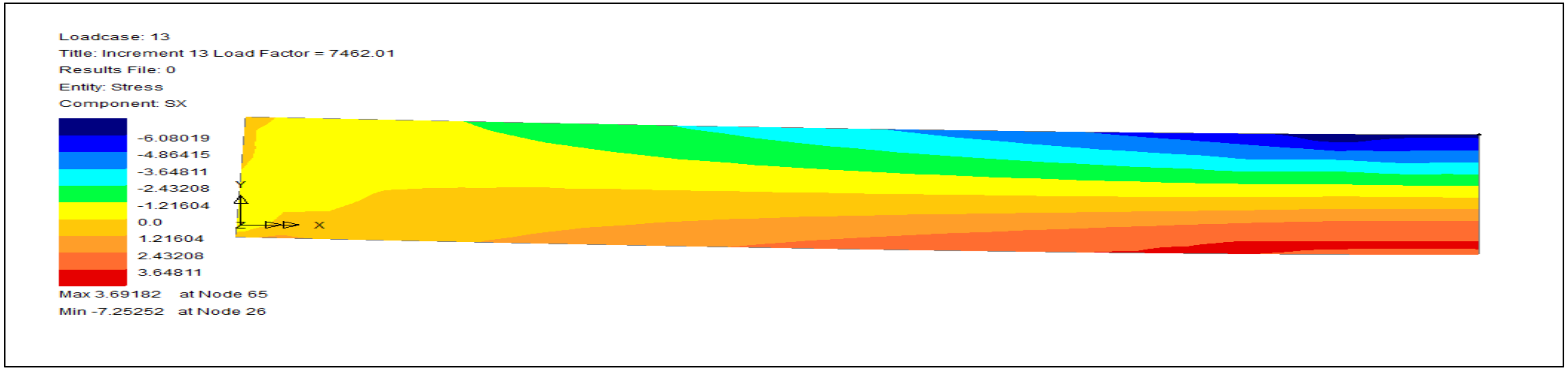


Figure 4.6 Stress SX Contour for load 7.46N (Beam SC545)

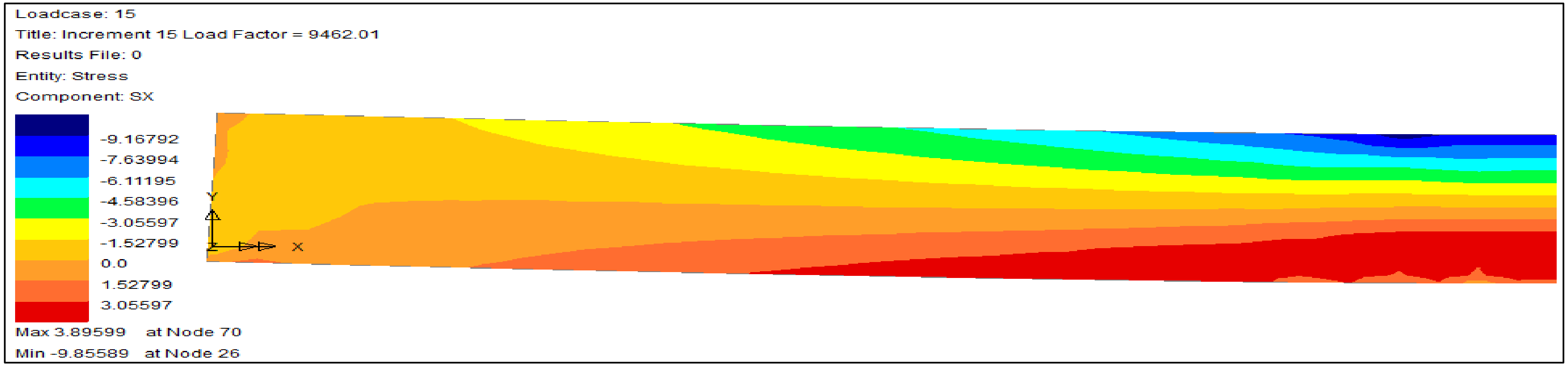


Figure 4.7 Stress SX Contour for load 9.46N (Beam SC545)

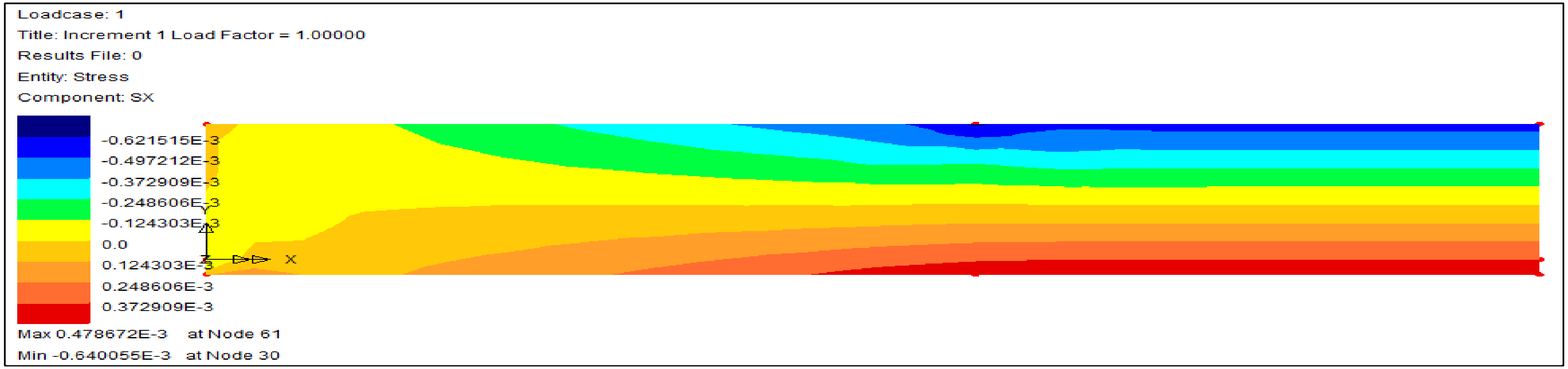


Figure 4.8 Stress SX Contour for load 12.46N (Beam SC545)

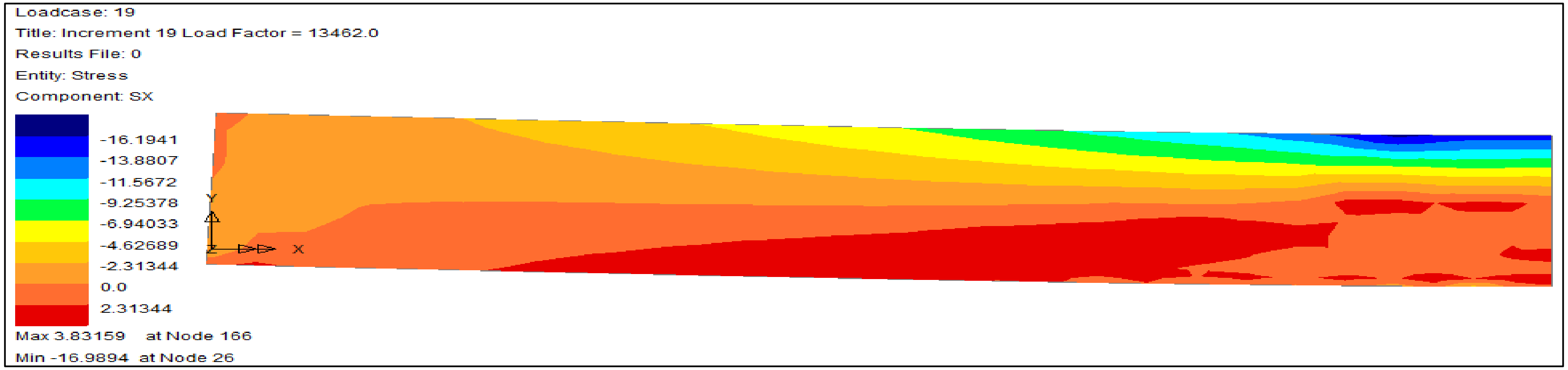


Figure 4.9 Stress SX Contour for load 13.46kN (Beam SC545)

Figure 4.4 until Figure 4.9 shows the stress contour for beam model specimen SC545 until it reaches its first crack value. For beam structure specimen SC545 the value of its first crack load is at 14.74 kN while for the beam model the value of the first crack load is at 13.46 kN. Generally we know that concrete structure is very strong in compression but very weak in tension. This is the reason why most of the concrete structures need to be reinforced with steel or fibre reinforcements. In this case, two bars of steel reinforcement with 16mm diameter size (Area: 402mm^2) were used.

Based on Figure 4.4 or before the start of the non-linear analysis, it can be seen that the area is about balanced since there is pressure applied on the model. The tensile areas were represented with the areas with reddish colour while compressive areas were represented with the areas prone to blue colour. Yellow and orange in colour was the zone with the most balanced forces.

After the load started to be applied to the beam model, changes were expected to occur in the stress contour. Tensile forces will be searching for the areas that were weak in tensile stress and thus leading to the failure of the structure. Figure 4.5 shows that the most reddish area or considered as the tensile stress happened to move up from the original positions which is the bottom part of the model specimens. As the bottom part of the concrete which is weak in tensile stress was applied load to it, the area will be transferring its stress to the reinforcement since it cannot handle the tensile stress applied to the bottom part.

From Figure 4.7 until Figure 4.9 shows the movement of the tensile stress in the model that happen after the load increment until it reaches its first crack value. In the stress contour shows that the compressive stress in beginning to lower down as the tensile stress is now took part in most of the areas in the beam. The areas that are still good in compressive forces are surely in the top part of the beam or specifically around the area where the point load is being applied.

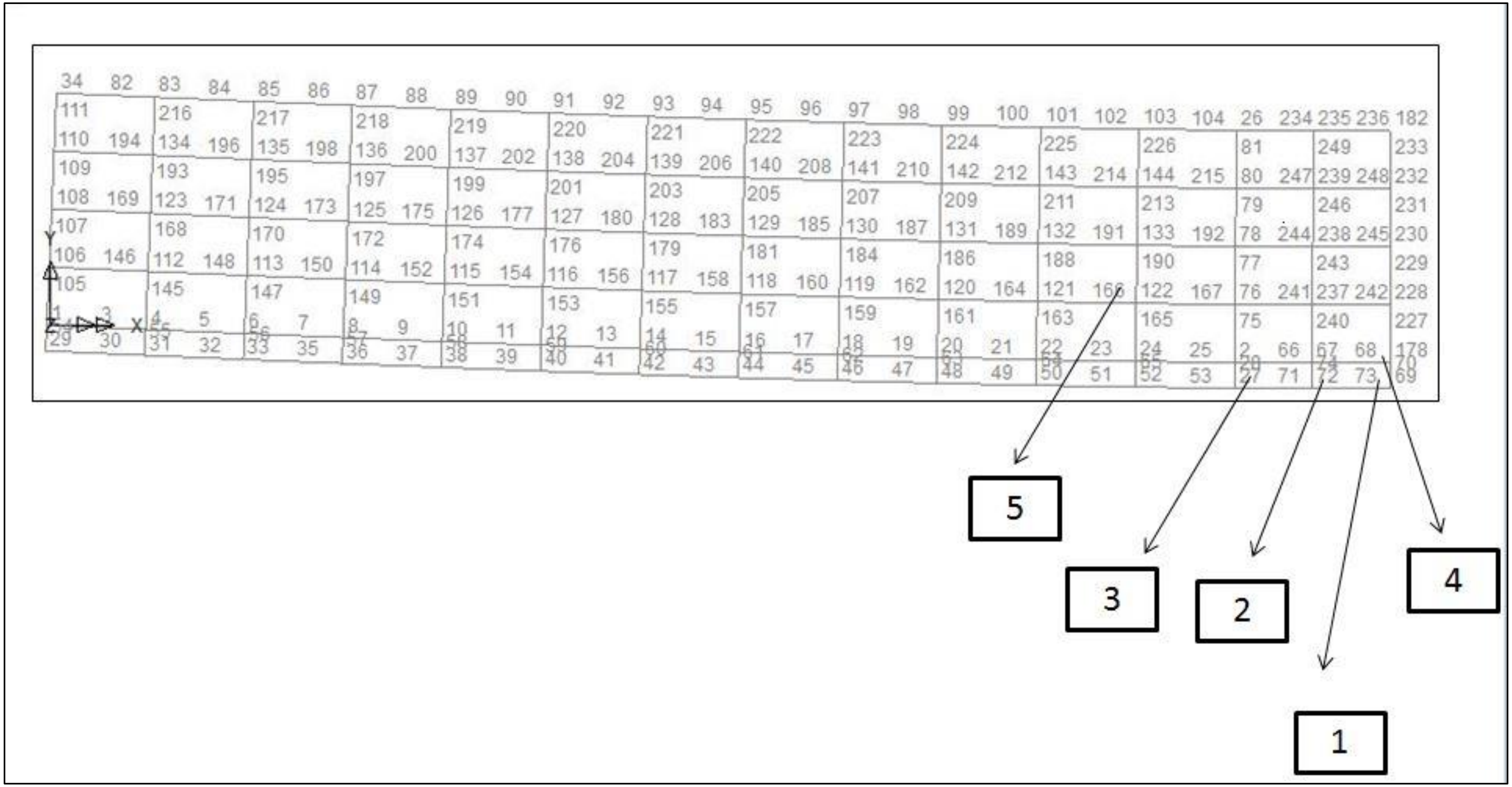


Figure 4.10 Maximum Stress Patterns for Every Load Increment (Beam Model SC545)

Table 4.3 Maximum stress at each load increment for SC545

Load Increment (kN)	**Node Number at Maximum Stress**	**Maximum Stress**
1.0	69	0.713E-3
4.462	69	3.2259
5.462	72	3.60478
6.462	27	3.68907
7.462	65	3.69182
8.462	70	3.71638
9.462	70	3.89599
10.462	70	4.00053
11.462	70	3.99817
12.462	70	3.82809
13.462	166	3.83159

Based on Figure 4.9 and Table 4.3, the figure shows the maximum stress patterns for beam model SC545. The stress patterns show that the first stress comes from node 69 when first load apply was 1N. The maximum stress maintain at the node 69 until load applied was at 4.46 kN until was changed to load 5.46 kN at node 72 and at load 6.46 kN at node 27.At load increment of 7.46 kN the maximum stresses happen at node 65 until it was maintained at the node 70 for both load 8.46 kN until 12.46 kN. At the first crack load, the maximum stress happened at the node 166, which above from the entire node said earlier.

By observing to Table 4.3 and Figure 4.10, the maximum stresses frequently happened between node 65 until node 70 which is at the bottom part of the model or at the center part of the beam. Since a_v/d for this beam is 5.45, theoretically said this beam is expected to fail due to flexural failure. As the beam was going to bend downward, surely it created many maximum stresses between the bottom centre parts of the beam.

It is difficult to say from this results that this beam will failed due to flexural failure, but if extension of load until it reaches its failure load, the beam are going to fail due to flexural failure with concrete crushing. At the first crack load tested to this model, the beam had shown the correct sign of failure because the maximum stresses were happened at the bending part of the model.

4.3.2 Beam Model for Specimen SC355

This beam is tested at a_v/d ratio of 3.55. According to the theoretical study, the beam will fail due to shear failure of the beam. Moreover, beam in this series does not have any shear reinforcement in it. Having no shear reinforcement will reduced the dowel action resistance capacity, caused a serious split to the concrete. If the theoretical study is correct, the diagonal shear crack with start from near the loading point of application to the end of support.

Figure 3.6 from Chapter 3 shows the node numbering for beam model for specimen SC355. The point load is applied at the node number 30, and the roller support is placed at node number 21. Node number 58 shows the bottom centre part of the beam or at the end of this model.

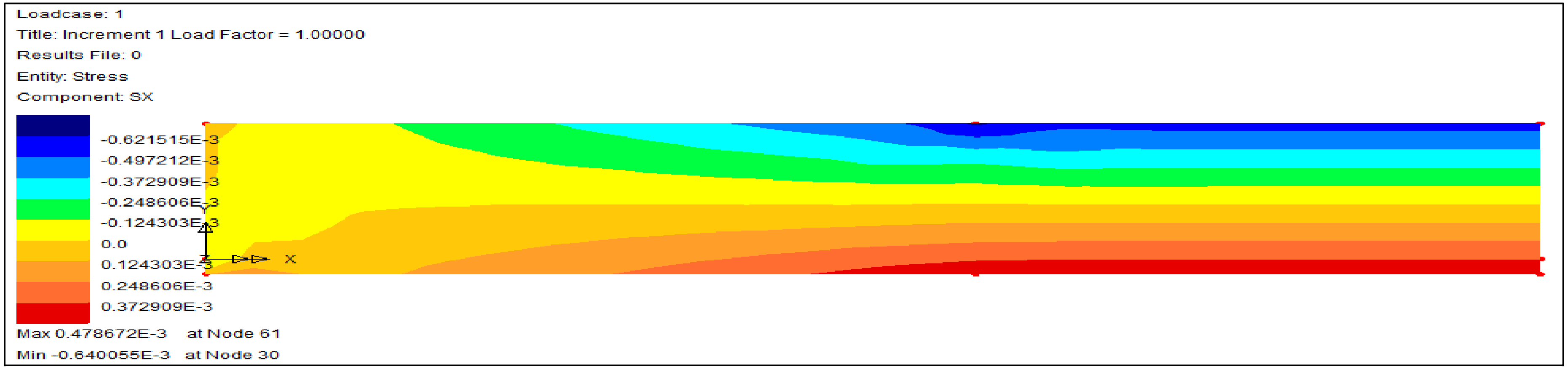


Figure 4.11 Stress SX Contour for load 1N (Beam SC355)

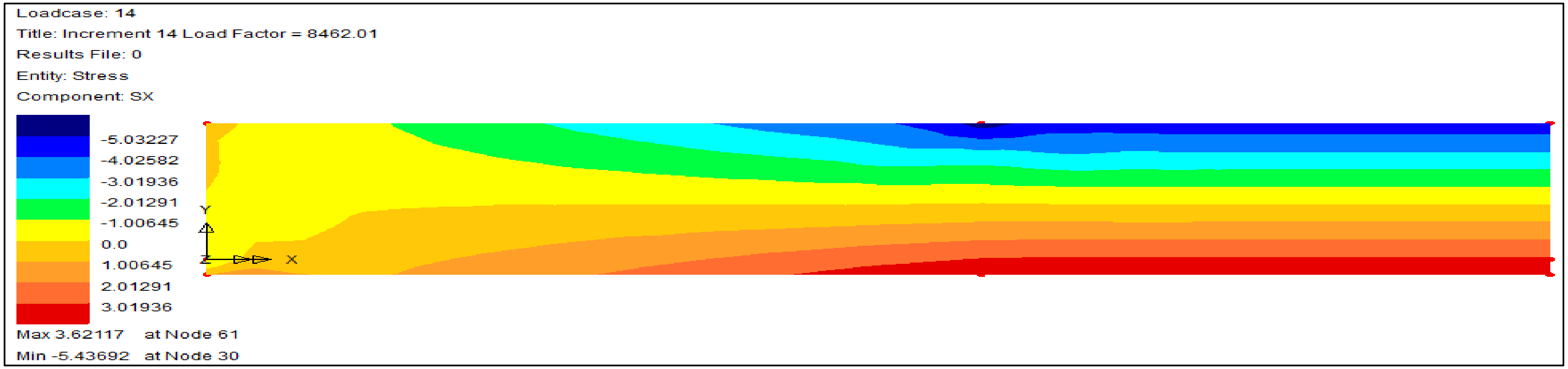


Figure 4.12 Stress SX Contour for load 8.46kN (Beam SC355)

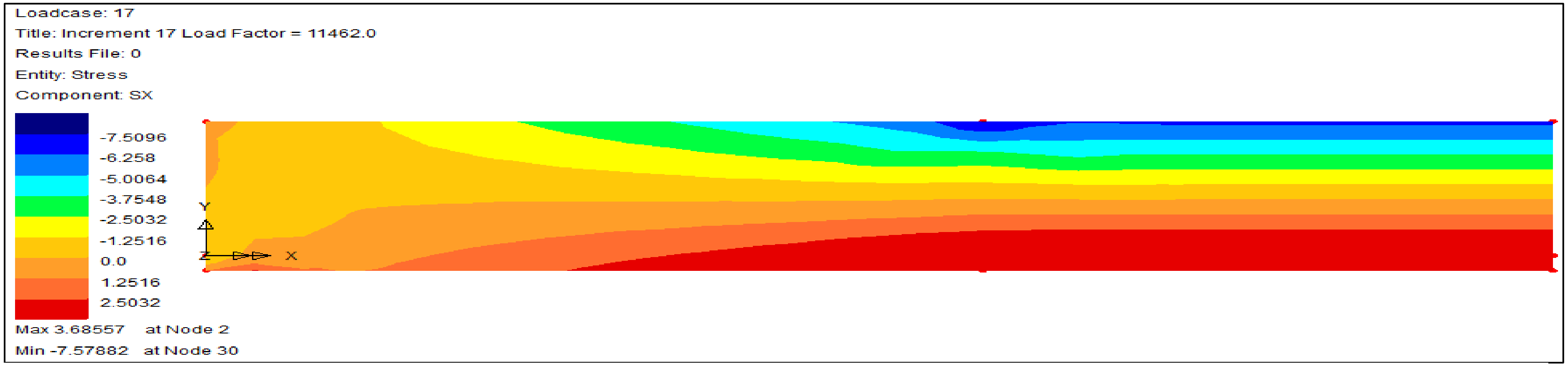


Figure 4.13 Stress SX Contour for load 11.46kN (Beam SC355)

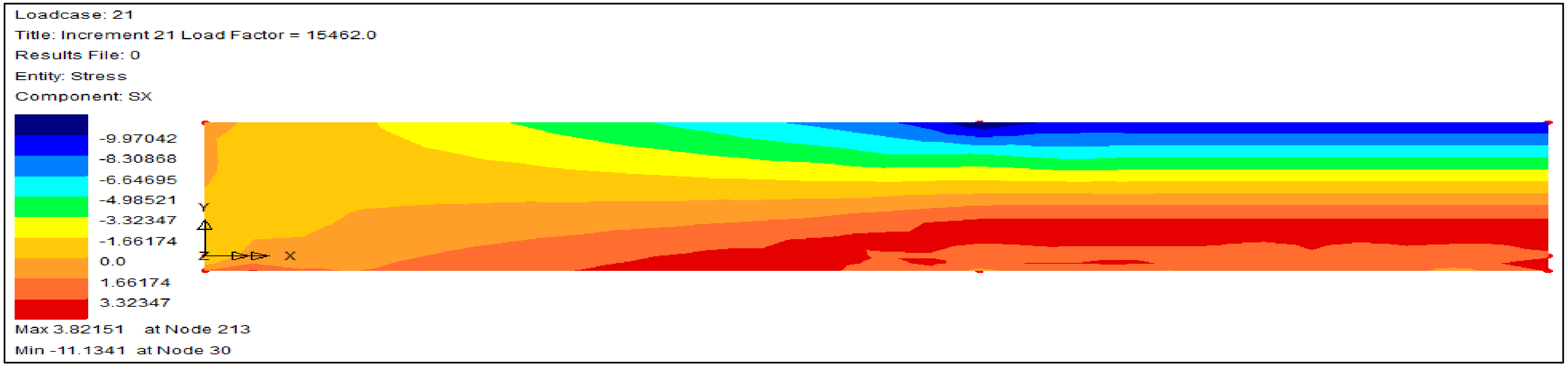


Figure 4.14 Stress SX Contour for load 15.46kN (Beam SC355)

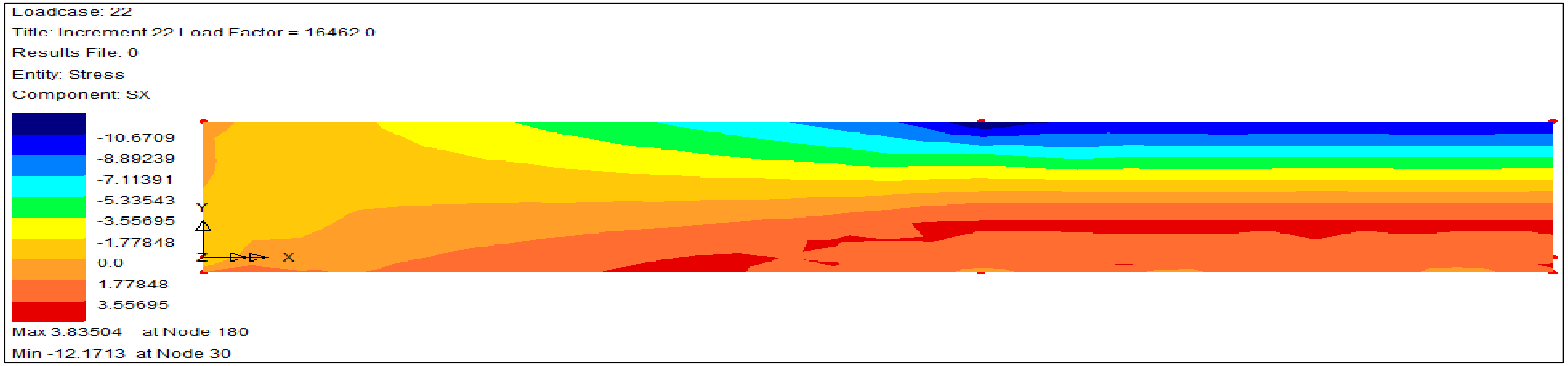


Figure 4.15 Stress SX Contour for load 16.46kN (Beam SC355)

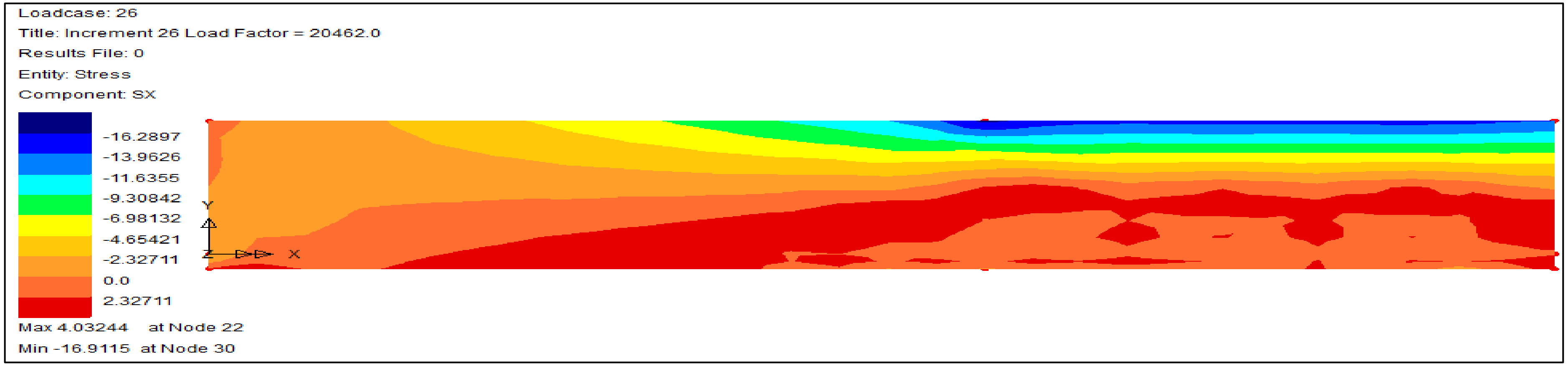


Figure 4.16 Stress SX Contour for load 20.46kN (Beam SC355)

Figure 4.11 until Figure 4.16 shows the stress contour for plane stress for beam model specimen SC355 until it reaches its first crack value. For beam structure specimen SC355 the value of its first crack load is at 22.61kN by Mohd Yunus (2012) while for the beam model the value of the first crack load is at 20.46. Generally we know that concrete structure is very strong in compression but very weak in tension. This is the reason why most of the concrete structures need to be reinforced with steel or fibre reinforcements. In this case, two bars of steel reinforcement with 16mm diameter size (Area: 402mm^2) were used.

Based on Figure 4.11 or before the start of the non-linear analysis, it can be seen that the area is about balanced since there is pressure applied on the model. The tensile areas were represented with the areas with reddish colour while compressive areas were represented with the areas prone to blue colour. Yellow and orange in colour was the zone with the most balanced forces.

Looking at the contour pattern, we can directly say that before the first crack values had reached, it takes a longer time a higher value of load before it actually starts to make a change in the stress contour. By looking at the first crack value also we can conclude that beam model specimen SC355 is more ductile than beam model specimen SC545. The stress area for beam SC355 also had extended since the position of the point load is now far to the support point.

The stress contour did not have any big difference if it were compared to the beam model SC545, but it was notable looking at Figure 4.15 where the most stress region is beginning to diminish. The stresses actually began to change its stress pattern from area around the point load to the support of the beam model. The maximum stress pattern will be discussed later in this chapter and more detailed explanation will be discussed for this behaviour.

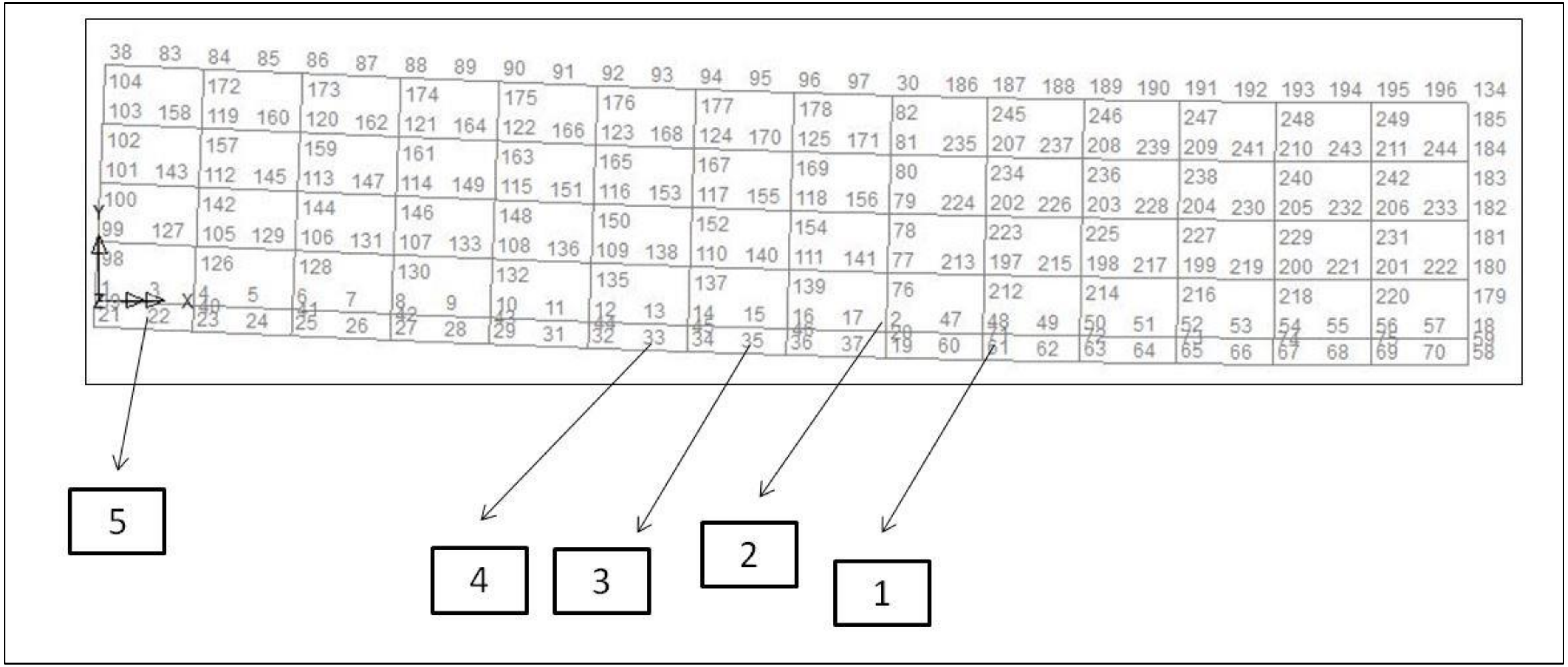


Figure 4.17 Maximum Stress Pattern for Every Load Increment (Beam Model SC355)

Table 4.4 Maximum stress at each load increment for SC355

Load Increment (kN)	**Node Number at Maximum Stress**	**Maximum Stress**
1.0	61	0.4679E-3
9.462	61	3.68874
10.462	71	3.69311
11.462	2	3.68557
12.462	35	3.68691
13.462	218	3.68868
14.462	33	3.69142
15.462	213	3.82151
16.462	180	3.83504
17.462	22	3.861
18.462	22	3.96408
19.462	22	4.0203
20.462	22	4.03244

Figure 4.17 and Table 4.4 shows the maximum stress pattern for beam model for specimen SC355. The maximum stress pattern for beam SC355 is different from beam SC545 as we can see that the maximum stresses frequently occurred at the area around the point load of the beam for SC355. The maximum stresses pattern also is quite unpredictable and scattered around but still around the area of the point load.

At the start of applying the load, the maximum stress happened at the node 61 which is beside the point load line but at the bottom part of the beam. The maximum stresses keep happened at the node 61 until the load increment reaches 9.46 kN value. From then, as the load is increased, the nodes kept changing from 71, 2, 33, 218, 33, 213 and 180.

At the next load which 17.46 kN, the load drastically moved to the node 22 which at the support of the beam. This is related to Figure 4.15 where at the load of 16.46 kN the stresses were slightly diminishes and actually at the value of 17.46 kN the stresses were completely diminished. This is the point where the maximum stress change from the center part of the beam to the end part of the beam which is the point load. This is due to a_v/d for this beam is 3.55 which this beam had been predicted to fail in shear failure. The stress will then surely find the area in the beam which is weak in shear thus producing the shear failure. It can be note that from 17.46 kN the node changes to node 22 for the maximum stress and the node is maintained until the beam reaches its first crack value which at 20.46 kN.

It is difficult to say from this results that this beam will failed due to shear failure, but if extension of load until it reaches its failure load, the beam are going to fail due to shear failure with diagonal crack. At the first crack load tested to this model, the beam had shown the correct sign of failure because the maximum stresses were happened at the support of the beam.

4.4 Concluding Remarks

From the observation of the maximum stress pattern, we can see the behaviour of both of the beam before it reaches its first crack load. It can be seen the nodes kept changing before it finally steady before reaching the first crack load value. There are only a little bit differences recorded between experimental results value by Mohd Yunus(2012) and the results from LUSAS analysis. Both of the beams shows the sign of correct stress pattern according to the beam failure tested in the experiment.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Introduction

The results presented have provided a very important contribution towards the understanding and knowledge on the behaviour of reinforced concrete beams. The structural behaviour of normal reinforced concrete beams which were provided with longitudinal steel reinforcement and other aspect as required by theory was studied. The beams were modelled as to be compared with the previous thesis done by Mohd Yunus (2012). Two beams that were chosen in the thesis were tested under the point load with two ranges of shear span to effective depth (a_v/d) ratio which are 5.45 and 3.55. From observations and theoretical studies, the beams were subjected to flexural and shear load, a lot of information especially on the structural pattern including the load-deflection behaviour, concrete and steel strain due to first crack load, mode of failure and others have been obtained as in the findings.

5.2 Conclusions

The conclusions on the nonlinear analysis of beams using LUSAS that can be drawn from the studies can be listed as follows;

i. Both of the beams chosen from the experimental study by Mohd Yunus (2012) were modeled in the LUSAS using the appropriate meshing and suitable attributes in order to match the experimental data. The nonlinear analysis was done until the first crack load to both of the beam and the results were drawn out later.

ii. The first beam tested from the experimental study by Mohd Yunus (2012) which is SC545 is expected to fail due to flexural failure. The tension zone of the concrete beam model yield and it will be followed by the concrete crushing in the compression zone. The maximum stress pattern that were drawn earlier shows the correct signs of flexural failure where the maximum stresses occurred at the nodes that is situated at the bottom-center of the beam. The maximum stresses also shows the signs of moving to the compression zone of the concrete beam.

iii. The second beam tested from the experimental study by Mohd Yunus (2012) which is SC355 is expected to fail due to shear failure with diagonal crack. The beam model will failed under the typical brittle shear manner, with diagonal crack occurred from the point load to the end support. According to the stress pattern drawn out earlier, it shows the maximum stress started to show in the area of point load before it was drastically changed to the support point. This shows the correct sign of shear failure due to the first crack value.

iv. Results from both of the beam models were compared to the original experimental results by Mohd Yunus (2012) for verification purposes. From the graph of load-displacement and value of concrete and steel strain, no big difference were recorded in between both of the results thus concluding that the software is verified for finite element and nonlinear analysis testing.

5.3 Recommendations

All results that were presented in this thesis can be the steps towards the right direction of investigating the behaviour for plastic analysis on concrete beam. Nevertheless, the recommendations for further studies investigations are to ensure that the studies are continues and be broaden until it is perfectly proven. The lists are as follows.

i. The beams can be tested until the failure load so that we can actually see the overall stress pattern that happened in a concrete beam. Then, we can finally prove from a nonlinear analysis that the beam failed in the failure type based on its a_v/d value.

ii. The nonlinear analysis test can be carried out at other a_v/d ratios since it is the important parameter in shear failure. The test should be done so the result can be used to study on its stress behavior and compare it with the other stress pattern from other a_v/d ratios.

REFERENCES

Bashkar, K. & Varadan, T.K. (2009), *Theory of Isotropic/ Ortotrophic (An Introductory Primer)* (1st ed.). New Delhi (ID) : CRC Press

Heinle, E. (1997). The design and construction of building in visual concrete. In Heinle, E. & Bacher, M., *Building in Visual Concrete* (pp. 1-3). London : The Whitefriars

Henry Tan (2009). *Flow Theory of Plasticity* [PowerPoint slides]. Retrieved from http://homepages.abdn.ac.uk/h.tan/pages/teaching/plasticity/Experimental.pdf

Johnson, K. (2008). *Solid Mechanics Part II*. Retrieved from http://homepages.engineering.auckland.ac.nz/~pkel015/SolidMechanicsBooks/Part_II/08_Plasticity/08_Plasticity_01_Introduction.pdf

Mohd Yunus Ishak (2012). *Shear Behaviour of Reinforced Concrete Beams Strengthened With Carbon Fibre Reinforced Polymer Fabrics.* Doctor Philosophy, Universiti Teknologi Malaysia.

Plumier,A. (2000). General report on local ductility. *Journal of Constructional Steel Research 55 (2000*) 91-107

Rajasekaran, S. (2003). *Finite Element Analysis In Engineering Design.* (1st ed.) New Delhi (ID)

Robinson, J. (2001), *Finite Element Methods In The Commercial Environment*. (3rd ed.) Great Britain (UK) : The Pitman Press, Bath

Spencer, G.C. (1968), *Nonlinear Elasticity*. (1st ed.) Netherlands. : CRC Press

Timoshenko, S.P. & Goodier, J.N. (1987), *A McGraw Hill Classic Textbook Reissue : Theory of Elasticity.* (3rd ed.) United States

Yongtao W,Liming K, Sizhuo S, Yuxing D, Zhong G, Zhirui Y, Qigang Z. (2012). Finite Element Analysis on von Mises stress distribution of Si DSP. *Material Science in Semiconductor Processing 16 (2013)* 165-170